

Reiner Anderl  
Peter Binde

# Simulations with NX

**Kinematics, FEA, CFD, EM and Data Management.  
With numerous examples of NX 9**

Contents, Preface, Sample Pages, Index

Reiner Anderl, Peter Binde

## **Simulations with NX**



Reiner Anderl  
Peter Binde

# Simulations with NX

Kinematics, FEA, CFD, EM and Data Management.  
With numerous examples of NX 9



HANSER

The authors:

Prof. Dr.-Ing. Reiner Anderl, Technische Universität Darmstadt

Peter Binde, Dr. Binde Ingenieure, Design & Engineering GmbH, Wiesbaden

Translated by the authors with the help of Dimitri Albert, Jan Helge Bøhn, Martin Geyer and  
Andreas Rauschnabel



Distributed in North and South America by Hanser Publications  
6915 Valley Avenue, Cincinnati, Ohio 45244-3029, USA  
Fax: (513) 527-8801  
Phone: (513) 527-8977  
[www.hanserpublications.com](http://www.hanserpublications.com)

Distributed in all other countries by Carl Hanser Verlag  
Postfach 86 04 20, 81631 Munich, Germany  
Fax: +49 (89) 98 48 09  
[www.hanser-fachbuch.de](http://www.hanser-fachbuch.de)

The use of general descriptive names, trademarks, etc., in this publication, even if the former are not especially identified, is not to be taken as a sign that such names, as understood by the Trade Marks and Merchandise Marks Act, may accordingly be used freely by anyone.

While the advice and information in this book are believed to be true and accurate at the date of going to press, neither the authors nor the editors nor the publisher can accept any legal responsibility for any errors or omissions that may be made. The publisher makes no warranty, express or implied, with respect to the material contained herein.

Cataloging-in-Publication Data is on file with the Library of Congress.

Bibliografische Information der deutschen Bibliothek:

Die Deutsche Bibliothek verzeichnet diese Publikation in der Deutschen Nationalbibliografie; detaillierte bibliografische Daten sind im Internet über <http://dnb.d-nb.de> abrufbar.

All rights reserved. No part of this book may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying or by any information storage and retrieval system, without permission in writing from the publisher.

© Carl Hanser Verlag, Munich 2014  
Copy editing: Jürgen Dubau, Jan Helge Bøhn  
Production Management: Andrea Reffke  
Coverconcept: Marc Müller-Bremer, [www.rebranding.de](http://www.rebranding.de), Munich  
Coverdesign: Stephan Rönigk  
Typeset, printed and bound by Kösel, Krugzell  
Printed in Germany  
ISBN 978-1-56990-479-4  
E-Book ISBN 978-1-56990-480-0

# Contents

<b>Preface</b>	1
<b>1 Introduction</b>	3
1.1 Learning Tasks, Learning Objectives, and Important Prerequisites for Working with the Book	5
1.2 Work Environments	7
1.3 Working with the Book	8
<b>2 Motion-Simulation (Multibody Dynamics)</b>	11
2.1 Introduction and Theory	11
2.1.1 Simulation Methods	12
2.1.2 Restrictions	14
2.1.3 Classifications of MBD	14
2.2 Learning Tasks on Kinematics	15
2.2.1 Steering Gear	15
2.2.2 Top-down Development of the Steering Lever Kinematics	33
2.2.3 Collision Check on Overall Model of the Steering System	50
2.3 Learning Tasks on Dynamics	59
2.3.1 Drop Test on Vehicle Wheel	59
2.4 Learning Tasks on Co-Simulation	68
2.4.1 Balancing a Pendulum	68
<b>3 Design-Simulation FEM (Nastran)</b>	79
3.1 Introduction and Theory	80
3.1.1 Linear Statics	81
3.1.2 Nonlinear Effects	83
3.1.3 Influence of the Mesh Fineness	85
3.1.4 Singularities	86
3.1.5 Eigenfrequencies	87
3.1.6 Heat Transfer	89
3.1.7 Linear Buckling	90

<b>3.2</b>	<b>Learning Tasks on Design Simulation</b>	90
3.2.1	Notch Stress at the Steering Lever (Sol101)	91
3.2.2	Temperature Field in a Rocket (Sol153)	139
<b>4</b>	<b>Advanced Simulation (FEM)</b>	149
<b>4.1</b>	<b>Introduction</b>	150
4.1.1	Sol 101: Linear Static and Contact	151
4.1.2	Sol 103: Natural Frequencies	151
4.1.3	Sol 106: Nonlinear Static	152
4.1.4	Sol 601/701: Advanced Nonlinear	152
<b>4.2</b>	<b>Learning Tasks on Linear Analysis and Contact (Sol 101/103)</b>	154
4.2.1	Stiffness of the Vehicle Frame	154
4.2.2	Size and Calculation of a Coil Spring	185
4.2.3	Natural Frequencies of the Vehicle Frame	199
4.2.4	Clamping Seat Analysis on the Wing Lever with Contact	207
<b>4.3</b>	<b>Learning Tasks Basic Non-Linear Analysis (Sol 106)</b>	229
4.3.1	Analysis of the Leaf Spring with Large Deformation	229
4.3.2	Plastic Deformation of the Brake Pedal	239
<b>4.4</b>	<b>Learning Tasks Advanced Nonlinear (Sol 601)</b>	249
4.4.1	Snap Hook with Contact and Large Deformation	249
<b>5</b>	<b>Advanced Simulation (CFD)</b>	271
<b>5.1</b>	<b>Principle of Numerical Flow Analysis</b>	272
<b>5.2</b>	<b>Learning Tasks (NX-Flow)</b>	273
5.2.1	Flow Behavior and Lift Forces at a Wing Profile	273
<b>6</b>	<b>Advanced Simulation (EM)</b>	297
<b>6.1</b>	<b>Principles of Electromagnetic Analysis</b>	298
6.1.1	Electromagnetic Models	299
6.1.2	Maxwell Equations	300
6.1.3	Material Equations	302
6.1.4	Model Selection	303
6.1.5	Electrostatics	306
6.1.6	Electrokinetics	306
6.1.7	Electrodynamics	306
6.1.8	Magnetostatics	307
6.1.9	Magnetodynamics	307
6.1.10	Full Wave (High Frequency)	307
<b>6.2</b>	<b>Installation and Licensing</b>	308

<b>6.3</b>	<b>Learning Tasks (EM)</b>	310
6.3.1	Coil with Core, Axisymmetric	311
6.3.2	Coil with Core, 3D	326
6.3.3	Electric Motor	330
<b>7</b>	<b>Management of Analysis and Simulation Data</b>	351
<b>7.1</b>	<b>Introduction and Theory</b>	351
7.1.1	CAD/CAE Integration Issues	351
7.1.2	Solutions with Teamcenter for Simulation	352
<b>7.2</b>	<b>Learning Tasks on Teamcenter for Simulation</b>	354
7.2.1	Carrying out an NX CAE Analysis in Teamcenter	355
7.2.2	Which CAD Model Belongs to which FEM Model?	362
7.2.3	Creating Revisions	365
<b>8</b>	<b>Manual Analysis of a FEM Example</b>	371
<b>8.1</b>	<b>Task Formulation</b>	371
<b>8.2</b>	<b>Idealization and Choice of a Theory</b>	372
<b>8.3</b>	<b>Analytical Solution</b>	372
<b>8.4</b>	<b>Space Discretization for FEM</b>	373
<b>8.5</b>	<b>Setting up and Solving the FEA System of Equations</b>	374
<b>8.6</b>	<b>Analytical Solution Compared with Solution from FEA</b>	376
	<b>Bibliography</b>	379
	<b>Index</b>	383



# 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.

Since 2003 Technische Universität Darmstadt has been selected and approved as PACE university and has become a 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 Corp. (in Germany Adam Opel GmbH). PACE is driven by General Motors Corp., Autodesk, HP (Hewlett Packard), Siemens, Oracle, and further well acknowledged companies of the virtual product development branch ([www.pacepartners.org](http://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](http://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](http://www.dik.maschinenbau.tu-darmstadt.de)).

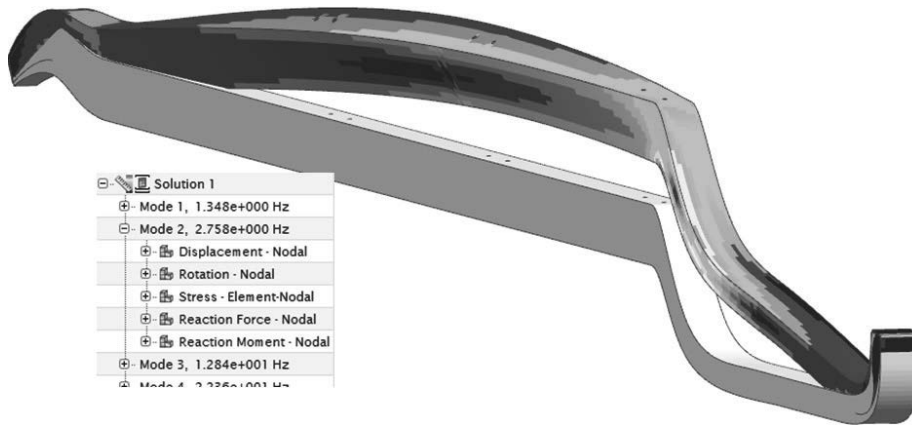
We thank very much Mr. Haiko Klause for his support to chapter 7 and Mr. Andreas Rauschnabel for his contribution to the Motion and FEM examples for Version 9 of the CAD system NX™. Furthermore we are grateful for the support of Carl Hanser Verlag, mainly Mrs. Julia Stepp. A very special thank you is 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 also in English.

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

August 2014

*Prof. Dr.-Ing. Reiner Anderl*

*Dr.-Ing. Peter Binde*



In modal analysis, the stress profile can be specified only qualitatively.

A colored version of this figure is available at [www.drbinde.de/index.php/en/203](http://www.drbinde.de/index.php/en/203)

The result should look as shown in the figure above. In this case, it is recommended to switch off the legend, so that the arbitrary numerical values are not visible.

✚ Save the files and close them.

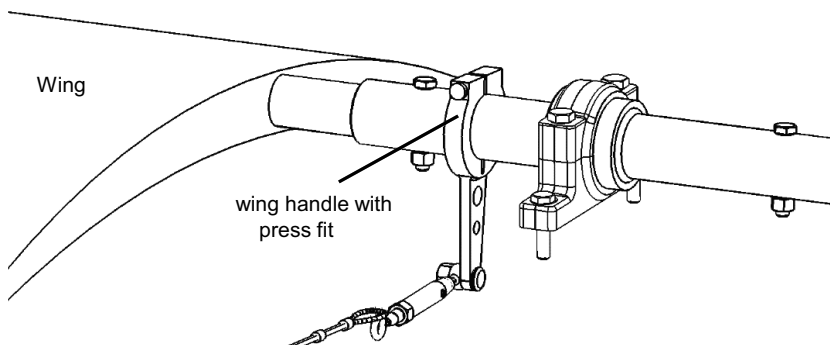
Thus this learning task is completed.

#### 4.2.4 Clamping Seat Analysis on the Wing Lever with Contact

For clamped connections (also press fits or similar) there exist analytical formulas in engineering handbooks [RoloffMatek]. However, these machine elements are usually CAD modeled by designers and integrated with in components and assemblies.

Analytical formulas no longer help.

This leads to complex geometry being used – with the result that analytical design formulas from the engineering manuals provide only very limited meaningful results. Therefore, analysis of clamped connections and similar elements have become an important application for FEA. The methodology for this will be explained in this learning task.



Clamping seats, clamping elements or press fits are usually calculated in FEA with non-linear contact.

The focus of this learning task is the use of the non-linear contact, which is required in such applications, but also in many others. Thus, in this learning task, the need, the operation, the adjustment parameters, and finally the application of contact in a typical example will be shown. The methodology presented here can be applied to other contact examples in a very similar way.

#### 4.2.4.1 Task






To achieve the required transferable torque, a screw force of 2500 N is to be reached on the clamping lever of the wing seat. The question is whether at this load, the permissible surface pressure in the contact area or the allowable stress in the rest of the geometry will exceed. Therefore, a stress analysis is to be performed, on the basis of which occurring surface pressures and stresses can be estimated.

#### 4.2.4.2 Need for Non-Linear Contact and Alternatives

Non-linear contact is always necessary if in reality during deformation impact, lifting, sliding or rubbing surfaces between bodies occurs and if these effects must be modeled in detail in the simulation.

So the non-linear contact is often important for assemblies analyses. Theoretically, non-linear contact could be calculated between all parts of an assembly, because it could well be that somewhere a touch occurs. But this should be avoided, because contact analysis leads to clearly higher computational times, and because even further difficulties may occur, as shown in this example. Therefore, it is very important that the FE analyst thinks ahead and decides sensible, where to apply non-linear contact analysis and where maybe it is sufficient to use simplified types of connection. Frequently, it is even sufficient to generate fixed connections between components.




Such fixed component connections are simply linear in nature, i.e., they are defined a priori, and the parts always remain in conjunction even during the deformation. NX functions for such types of solid compounds are:

-  *Surface to Surface Gluing* for the flat connection of two bodies.
-  *Edge to Surface Gluing* for connecting of example shell elements to volume elements.
-  *Edge to Edge Gluing* for connecting at two edges.
-  *Mesh Mating Condition* for the alignment or merging of nodes of two bodies at an area.
-  *1D-Connection* with *RBE2*-elements (for example, a point with an area over rigid coupling elements)

Such cases have already been covered in previous examples. The functions which are available in the NX system for non-linear contact in the Nastran solution 101 for this are:

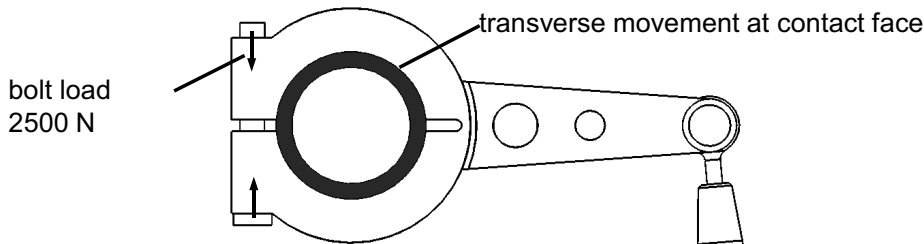
Contact analyses lead to significantly higher computation times.

Overview of linear functions for connections.

-  *Surface to Surface Contact* for sheet-like contact situations.
-  *Edge-to-Edge Contact*
-  *1D-Connection with CGAP-elements*, which can be defined, for example, from one edge to another edge.

Summary table of non-linear functions for contacts.

The press fit is an example in which two parts slide or rub on each other. Perhaps there would even be an initial gap between the contact surfaces, or perhaps there will, due to the load, arise a gap somewhere. A simple fixed connection at the contact surface, such as the functions Glue or Mesh-Mating-Condition allow, is not effective, when stresses at the contact surface shall be determined with good accuracy, because with the gluing all transverse movements of the parts to each other are suppressed, which are possible in reality. The simple bonding of the parts would therefore lead to unrealistic transverse stresses. For this reason, the non-linear contact in the case of the clamping seat useful.



With a clamping the two parts deform. The contact forces between the parts change during deformation. In addition, the surfaces slide on each other.

A possible alternative to the non-linear contact could be the function *Surface to Surface Gluing* if it is adjusted such that it possesses in the tangential direction – i. e. the sliding direction – a small and in the normal direction a large stiffness. Adjusted this way it corresponds to a sliding connection. This is possible because in this function, a separate stiffness definition of the two directions is possible.

The glue function, in some cases, can replace the non-linear contact.

#### 4.2.4.3 Operation of Nonlinear Contact

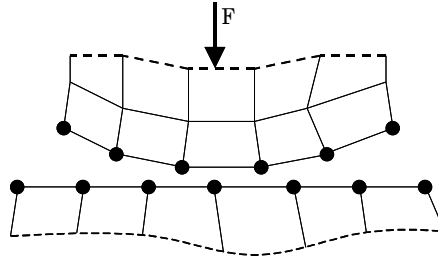
In the linear FE analysis stiffness and boundary conditions are determined only once from the component and added to the stiffness matrix and the load and displacement vector. In contrast, in the analysis with non-linear contact, it is checked again and again during the deformation, whether penetration of contact surface pairs happens. If there is such a penetration, opposing forces must be formed, which counteract the penetration.

To realize this, it is necessary that the external loads are applied in small steps and examined after each step, whether there appear penetrations in the potential contact area. Such penetrations can easily be found by checking geometric positions of corresponding nodes.

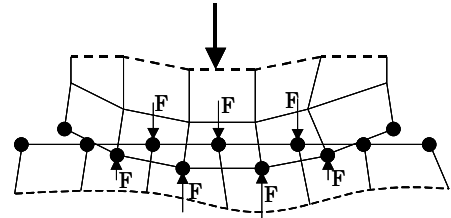
The contact leads to iterations.

Each of these load steps is realized by a linear finite element analysis, which proceeds much like usual. The only difference is that the load steps do not come from the relaxed body, but always include the preload of the previously already loaded body.

In the outer iteration (status loop) external forces are applied piecewise.



outer iteration (status loop)



inner iteration (force loop)

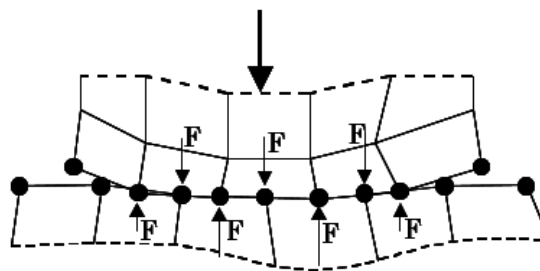
As long as no penetrations occur, the load is simply increased until the desired full load reaches at the end or until it comes to a penetration of nodes. The progressively increasing load and finding of contact nodes is called *outer iteration* or *status loop*. When it comes to the first node penetration, an algorithm starts that we call the *inner iteration* or *force loop*. This algorithm ensures that the nodes are pushed back so that they adjacent flat and realistically to each other at the end.

In the second iteration (force loop) contact restoring forces are applied.

The retraction of the inner nodes in the iteration is obtained by restoring forces that are applied to the corresponding nodes. So there are following linear FE analyses performed in which now additionally these restoring forces are inserted. The restoring forces can be estimated only roughly by the software using the current penetration of a node and multiplying it by a given stiffness value.

With such estimated restoring forces there will surely not immediately be achieved, that the contact nodes remain lying exactly on a surface. The restoring forces are assumed rather too large or too small. Therefore, restoring forces are recalculated based on the resulting node overlap and applied in the next linear FE analysis. Now, the overlap should have become smaller. This inner iteration is repeated until a convergence criterion has been reached.

After the contact restoring forces have been found correct, the next piece of external force can be applied.



the inner iteration has converged

After the inner iteration has converged, in the outer iteration, the next piece of load is applied, and the inner iteration begins again.

So the non-linear contact is a doubly nested iteration of each case linear FE analyses. Therefore, considerably larger computation times will arise. Non-linear contacts should therefore be used only when it is absolutely necessary.

#### 4.2.4.4 Loading the Assembly and Creating the File Structure

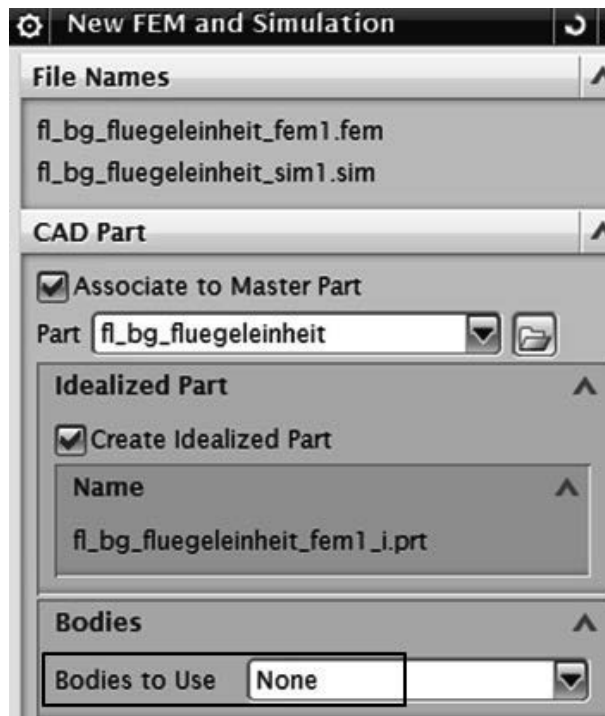
- ✚ To begin the task, you now load the assembly *fl\_bg\_fluegeleinheit.prt* from the RAK2 directory.

This assembly contains the two parts *fl\_fluegeltr\_klemmhebel* and *fl\_fluegel-traeger*. These are the clamping lever and the tube on which the clamp is to be realized.

- ✚ Now switch to the application *Advanced Simulation*.
- ✚ Create in the simulation navigator the structure for the simulation.

At first, a body is not yet included in the analysis.

Please note that not all bodies must be transformed into polygonal geometries, but only the geometry of the clamping lever and the tube. Moreover, these two bodies are supposed to be changed significantly in the idealized file. That's why it makes sense to initially create no polygon body at all and make this selection at a later date.



- ✚ Set the option *Bodies to Use* to *None*.

#### 4.2.4.5 Contact Specific Parameters in the Solution

After creating the file structure you will be prompted to define the solution. In the previous examples it was mostly confirmed here with **OK** and so the defaults were accepted. Also in the case of this example, the default settings can be accepted, too, but let's first explain some important settings for the control of the contact algorithm.

The default parameters should be changed only if problems appear.

In some cases, upon contact analyses, it is advisable to make changes to these parameters to reach acceptable results. We refer to the explanation on the contact algorithm of NX Nastran. Settings for other solver types must be found in the online help.

Below, we explain the meaning and give recommendations to some important contact parameters. These explanations are partially taken from the document NX Nastran User's Guide [nxn\_user], which belongs to the online help of NX Nastran. Some of the explanations are enriched by own experience of the authors.

The following figure shows the global contact parameters in which the declared parameters will be passed to the NX system menu. You can find this in the menu *Solution* in the *Case Control* register in *Create Object Modeling*.

The global contact parameters menu and the default values.

Description	
Card Name	BCTPARAM
<b>Primary Parameters</b>	
Contact Force Tolerance (CTOL)	0.01
Max Iterations Force Loop (MAXF)	10
Max Iterations Status Loop (MAXS)	20
Contact Changes for Convergence	Percentage of Active
Percentage of Active Contact Elements (NCHG)	0.02
Initial Penetration / gap (INIPENE)	Calculate from geome
Shell Thickness Offset (SHLTHK)	Include
Include Shell Element Z-Offset (ZOFFSET)	Include
<b>Secondary Parameters</b>	
Penalty Normal Direction (PENN)	
Penalty Tangential Direction (PENT)	
Penalty Factor Units (PENTYP)	1/Length
Contact Status (RESET)	Start from Previous
Refine Source Region Mesh (REFINE)	Use new algorithm
Number of Contact Evaluation Points (INTORD)	Increased Number
All Contact Elements Can Become Inactive (CSTRAT)	Yes
Export a Preview Bulk Data File (PREVIEW)	No

Stiffness values and recommendations for contact.

- *Penalty Normal Direction, PENN*: This value controls the stiffness in the normal direction when the contact surfaces run into overlap. Larger values lead generally to faster convergence and smaller maintaining overlap. However, too large values also mean that no convergence can be achieved. The default is a value of 10. This is not displayed, but internally stored by the software. If a value is entered, the default value is overridden for the analysis. There are following recommendations:
  - Stiffness values between 1000 and 10000 are suitable in most cases.
  - Stiffness values between 10 and 100 are suitable when the contact surfaces are identical but nodes do not match. Smaller values can smooth out irregularities of not matching meshes. If contact stresses are to be determined, we get the best results when matching meshes exist on both sides of the contact.

- Very small stiffness values, that is, values less than one, lead to very slow convergence and are not recommended.
- *Penalty Tangential Direction, PENT*: This value controls the convergence of the frictional forces when the friction is not zero. The value should generally be 10 to 100 times smaller than the Penalty Normal Direction.
- *Contact Force Tolerance, CTOL*: This is the limit for the contact-convergence tolerance (Euclid Norm). Once the contact convergence tolerance is less than this limit, the contact algorithm has converged. Termination criteria for contact
- *Max Iterations Force Loop, MAXF*: Here, the maximum number of iterations for the inner or force loop iteration can be set.
- *Max Iterations Status Loop, MAXS*: Specifying the maximum number of iterations for the outer or status loop iteration.
- *Percentage of Active Contact Elements, NCHG*: Hereby it can be achieved that contact analysis terminates, although not all contact elements have converged or become inactive. This makes sense in cases when coarser accuracy is acceptable or when it is recognized from the convergence history that the last contact elements converge very slowly or not at all. Such a behavior would indicate so-called contact-oscillations, i.e., the contact elements oscillate back and forth between the closed and open state. If this parameter is set such as 0.01, this means that 1% of the contact elements do not need to converge. If the contact has to stop sooner...
- *Shell Thickness Offset, SHLTHK*: The contact feature in NX Nastran can be used both for shells as well as the solid elements. This value is only relevant for shells and manages the control of the shell thickness in the contact area. If the option *Include* is used, the system assumes that the shell elements are at the center of the body, that is, the thickness  $t$  is assumed on both sides by  $t/2$ . However, if the shell elements represent the outer surface of the body, enter the option *Exclude*, then the thickness is ignored.
- *Contact Status, RESET*: If multiple load cases are calculated, with the default option *Start from Previous* it is achieved that for the analysis of subsequent load case, the final contact condition of the previous load case is utilized. This often makes sense to speed up contact computation.
- *Initial Penetration/Gap, INIPENE*: Unfortunately, there are often tiny initial penetrations of individual nodes of the contact surfaces. These are almost not visible at the FEM mesh, but result in unrealistic stress peaks on the contact surfaces. This parameter controls how such situations should be handled. The default setting *Calculate from Geometry* does not change anything at the given geometry. The option *Ignore Penetration/Use gaps* removes artificially penetration at positions where initial penetrations exist, therefore puts these areas exactly adjacent to one another. The third option *Set to Zero* sets all contact surface areas exactly to zero distance. This third option is very effective. In many practical contact tasks it has been found that good results could be achieved just by using this option. Invisible initial penetrations are often guilty of "dotted" stress results.

In the case of our example, all defaults can be accepted.

- ✚ Therefore, confirm with OK, and the solution will be generated.



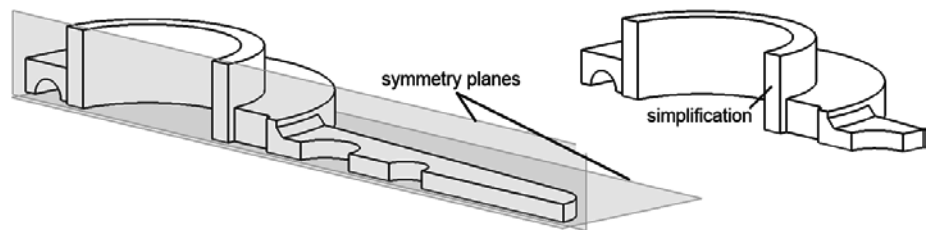
#### 4.2.4.6 Part 1: Rough Analysis with Tetrahedrons

In this first part of the task we want to come to a conclusion in a rapid manner. We just want to perform the most essential work. For that we accept a certain loss of precision. In the second part of the task we then pull out all the stops for a high-quality contact analysis.

#### 4.2.4.7 Geometry Simplifications for Symmetry

A mirror symmetry exists if on the left of a symmetry plane exactly the same deformation happens as on the right. In our task we have this condition twice. The following figure shows those two planes of symmetry and the already cut model, so as it should be after the operations in this section.

The model is simplified.  
Symmetry is used twice.



In most cases the symmetry is exploited in FEM tasks in order to save computing time and memory. Finally, it makes no sense to produce duplicate results if we already know that left of the plane the same result will come out as right. Symmetry should be exploited in any case, especially if it is anticipated that the analysis will take longer. At a contact task this is definitely the case.

To get to the double cut model presented, the following steps are to perform:

- ✚ Turn into the idealized file and create two wave links, one for the pipe and one for the clamp lever.
- ✚ Hide in the assembly navigator the entire assembly *fl\_bg\_fluegeleinheit*. The two wave-linked parts remain visible.
- ✚ Turn into the modeling application, create two *Datum Planes* at the corresponding planes of symmetry, and trim the two bodies at these planes.
- ✚ If you wish, you can also – as shown in the figure above right – trim a part of the lever, because this area has little impact on our results.
- ✚ Also, the pipe can be truncated a piece, so that only the part of interest is left.
- ✚ Switch back to the application *Simulation*.

#### 4.2.4.8 Add Polygon Geometries Subsequently

- ✚ Change to the FEM file.

The polygon bodies are not yet there or need to be replaced again.

You notice that there is no single polygon body in the FEM file. This is how it should be, because when generating the FEM file at *Bodies to Use* we have set the option to *No*. Now there is geometry ready prepared in the idealized file, so polygon geometry for meshing

shall be generated. To perform the subsequent generation of the polygon geometry, proceed as follows:

- ✚ Select from the simulation navigator on the uppermost node RMB and EDIT.
- ✚ Activate the switch *Edit Bodies to Use*. Now the idealized file is temporarily shown in the graph window.
- ✚ Switch the option *Bodies to Use* to SELECT.
- ✚ Select the two simplified bodies tube and clamping lever.
- ✚ Confirm with OK.

Now the two wave-linked and simplified geometries are shown in the FEM file and can be used.

#### 4.2.4.9 Material Properties

Both the pipe and the clamping lever should be made of common steel.

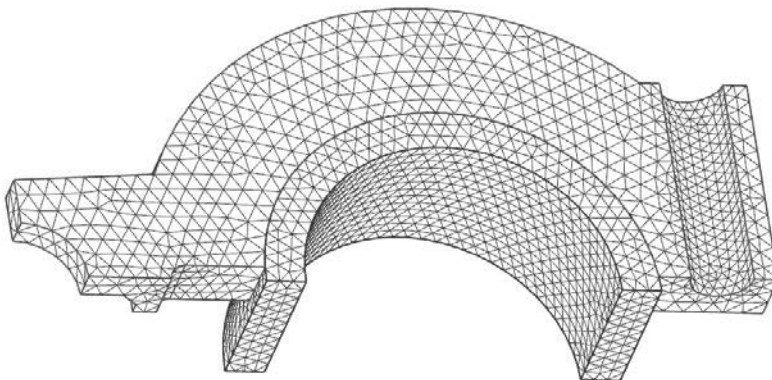
- ✚ Select the function *Assign Materials*.
- ✚ Search the library to find the material *Steel*.
- ✚ Select the two bodies.
- ✚ Confirm with OK.

#### 4.2.4.10 Meshing with Tetrahedrons

For a rough contact analysis a simple tetrahedral mesh is generated with refinement to half or one quarter of the value proposal. Proceed as follows:

- ✚ Select the function *3D Tetrahedral Mesh*.
- ✚ Leave the element type to the default *CTETRA10*.
- ✚ Select the clamping lever.
- ✚ Select the yellow flash for the proposal of element size.
- ✚ Divide this default value by two.
- ✚ Confirm with OK.
- ✚ Perform the same steps again with the shaft. Use here the default element size value divided by four.

A rough analysis should always be performed first.



This is a standard mesh: Tet10 elements and half or one quarter of the proposal value for the element size.

The result should look as shown in the figure above. As you see, the nodes at the contact point do not coincide. Although this mesh and contact analysis can be performed, but the accuracy of results in the contact area is limited.

#### 4.2.4.11 Symmetry and Further Boundary Conditions

- ✚ Switch to the simulation file.


The condition of symmetry is: No deformation perpendicular to the symmetry plane.

An appropriate boundary condition must now be applied to all cut surfaces of symmetry. There are two ways to choose from: First, the function *Symmetric Constraint* and secondly the general function *User Defined Constraint* could be used.

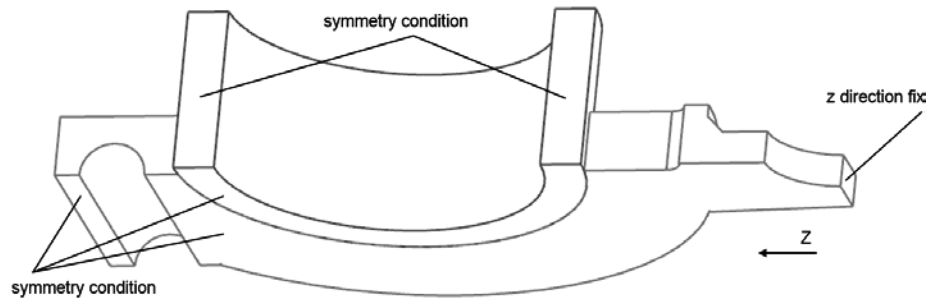
When using symmetric constraint condition you need not to worry about the directions. You should only avoid combining all planes of symmetry into a single condition. Rather, put always those faces together that belong to the same section – i.e., that face in the same direction.

When using the user-defined function, note that for symmetry conditions on volume elements this degree of freedom perpendicular to the cutting surface must always be fixed and all others must be free.

We want to use the symmetric constraint condition here:

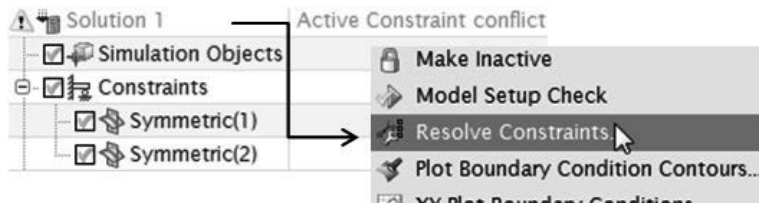
- ✚ Select the *Symmetric Constraint*  and select all faces that belong to the first symmetry-section. These are three faces.
- ✚ Confirm with OK.
- ✚ Create the same condition again on the three faces that belong to the second symmetry section.

These constraints prevent the movement of the clamping lever, but not that of the sleeve.

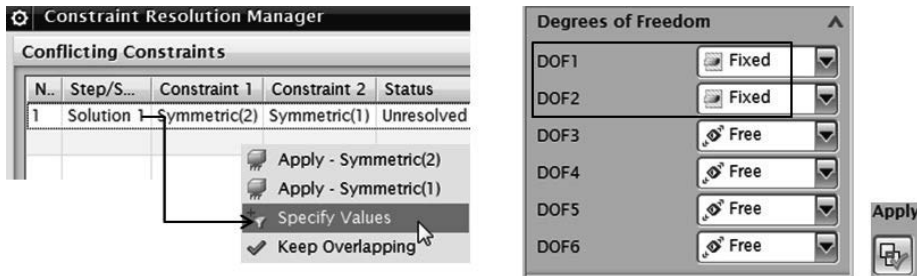


In the navigator on the solution node it can be seen that there are conflicts with the boundary conditions. The reason is that at some edges both symmetry conditions act. The system displays this conflict situation (see figure below) and the user should now intervene and define, how to proceed on these edges. There are several possibilities: The first (*Apply - Symmetric (1)*) or the second (*Apply - Symmetric (2)*) condition may be active or the degrees of freedom can be specified individually (*Specify Values*).

- ✚ After the two symmetry conditions have been created, switch to the navigator choose on the solution node with RMB the function *Resolve Constraints*.



- ✚ In the following dialog, use RMB on the displayed conflict and select the function *Specify Values*. Set the X and Y translations, i.e. DOF1 and DOF2, to fix and the Z translation to free.
- ✚ Click **APPLY** and **CLOSE**. The conflicts are solved now.



Moreover, movements in the Z-direction must be prevented. This is required even if under the given loads no movement in the Z-direction would occur. Static FEM analysis will always require that statically determined supports of the parts is given. Over-determined bearings are indeed allowed, but not under-determined.

- ✚ According to the figure above, you create a *User Defined Constraint* on the small sectional area, which fixes the Z-direction.
- ✚ Now we have created yet another overlap conflict in the solution. Solve this example by assigning the symmetry condition.

With this constraint set the clamping lever is mounted statically determined. The tube, however, still has a possibility of movement in the Z-direction. Preventing this will be our focus in the next section.

#### 4.2.4.12 Add Soft Springs for Static Determination

The requirement for static determination leads in particular in contact analysis in many cases to difficulties. The problem is the fact that in reality contacts contribute to determination, but in FE analysis, non-linear contacts act only in form of forces on the parts. However, the static determination must be achieved by merely boundary conditions in the FE system, therefore, the contact forces do not count.

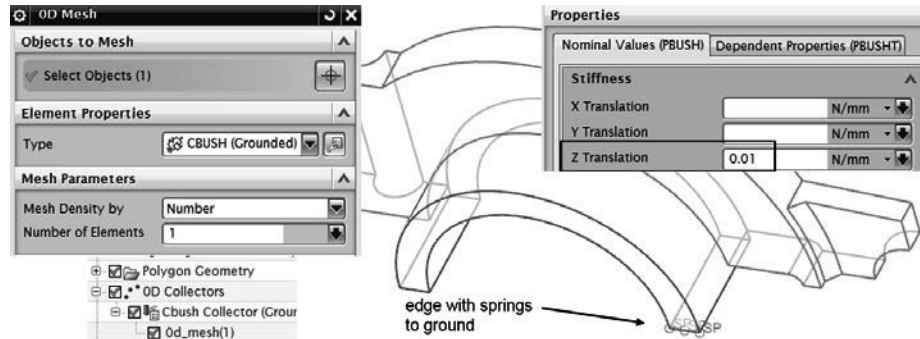
If we check in our task each part to its constraint situation, we note that the pipe is still undetermined in Z-direction. Now we have a typical problem: On the one hand, the tube needs a constraint in Z-direction, on the other hand, this constraint would distort the re-

The constraints may be over-determined, but not under-determined.

sult, because the position of the tube in the Z-direction will be determined by the contact forces.

This problem is typical in contact analysis. One possible solution is to support the tube with soft springs in the missing Z-direction. With such a soft spring the tube can adjust its position almost free. Nevertheless, there is statically determined support and the FE analysis can be performed.

Soft springs are a way to reach bearings without applying hard constraints.



Follow the steps for creating such a soft spring bearing as follows:

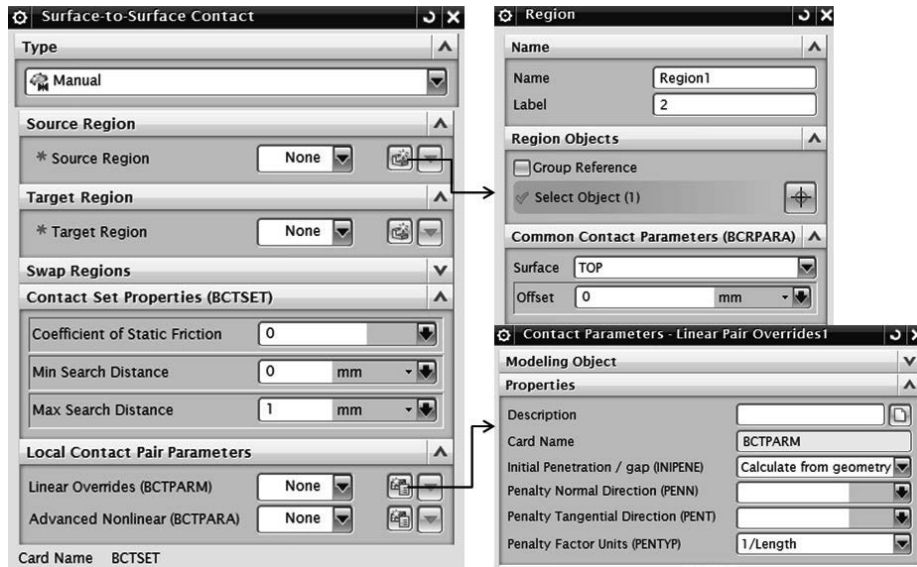
- 1 First, make the FEM file to the displayed part.
- 2 Generate a *OD-Mesh* of type *CBUSH (Grounded)* on an edge of the tube. It does not matter which edge you use.
- 3 In the navigator there is a mesh collector for the *OD Mesh*. Edit this and modify the contained *Physical*. Enter a small value for the stiffness in the Z-direction, for example, 0.01 N/mm.

The spring elements *CBUSH (Grounded)* used here have the property that they are automatically connected to the earth. This means that no further constraint in the SIM file is necessary.

#### 4.2.4.13 Definition of the Contact Area

The following is about the definition of contact areas, i.e. surface pairs, where during deformation will be examined for penetration and, where appropriate restoring forces will be applied. In addition, also the coefficient of friction is given here, to be in this area of contact. Go to the definition as follows:

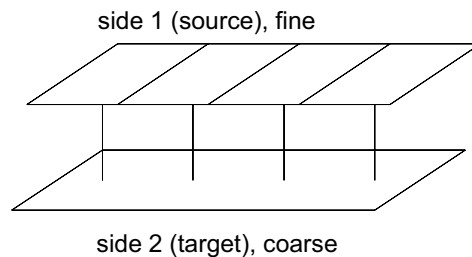
- 1 Make the simulation file to the displayed file.
- 2 Select the function *Surface to Surface Contact* in the functional group *Simulation Object Type*.



The contact is a non-linear function in the otherwise limited to linear static Nastran solution 101.

The contact requires the definition of a pair of surfaces. The first of the two partners we call *Source Region*, the second *Target Region*.

If meshes are of different element size on both sides of the contact, the following rule applies: The first side (Source Region) shall be the one with the smaller mesh size. The background is that the contact check is always performed starting from the first side. Using the nodes of the first side there are normal projections performed to the surfaces of the second side. So when the first side is meshed by smaller elements, the contact analysis can be performed with greater accuracy.



In case of different mesh sizes there is to pay attention to the order of selection: The first is to be the finer one.

In our example, the contact surface on the tube is meshed finer and should therefore be the starting area. So follow these steps:

1. Select at *Type* the *Manual* option.
2. Select the function *Create Region* in *Source Region*.
3. Select the cylindrical contact face on the tube.
4. Leave all settings in the menu *Region* at default and click **OK**.
5. Select at the *Target Region* the corresponding contact face on the clamping lever.

Now you have created a surface pair for the contact. Some further adjustments are still possible back in the main menu of the contact, which are sometimes very useful. These include:

The contact parameters at a glance.

- *Coefficient of Static Friction*: Here the coefficient of friction is given, which is responsible for ensuring that shear forces are transmitted from one body to another. These forces are calculated according to Coulomb's friction law using the pressure force.
- *Min./Max. Search Distance*: The contact algorithm will search for contact only in a range between these two limits. For the default setting zero and one, this means that analysis of the contact is performed only where the contact faces have smaller distance than 1 mm.

If press fits have oversize...

- The setting *Offset* that is available in the *Region* menu can be used to artificially move the contact surface a little. This corresponds approximately to an additional layer of material in the contact region. This could represent, for example, a layer of paint to be considered for the contact that is not specifically FE modeled. Even so, an initial penetration of the contact surfaces could be modeled this way, for example, as it is required in the analysis of press fits with interference.

For shell elements, there are particularities.

- *Surface*: This setting from the *Region* menu is only relevant if shell elements are in the contact area and moreover if the shell elements are defined as *Source* region. If this is the case, the contact checks are normally executed into top side direction of the shell elements. However, if this is the wrong side, there can be switched over to the bottom side using this option.
- Finally, there are the settings in *Linear Overrides (BCTPARM)*. Here some of the global contact settings can be overridden. This is useful if multiple contacts are in the model and different settings shall be used.

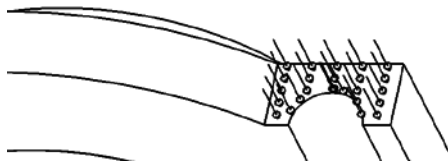
✚ Confirm with OK, so the contact function is generated.

#### 4.2.4.14 Generation of the Bolt Load

The last thing missing is the bolt load of 2500 N. According to the figure below, this can, for example, be applied to the supporting face of the screw head. Keep in mind that the applied bolt load in the FEM model is only half due to symmetry.


- ✚ According to the figure, create a force of 1250 N in the positive y-direction on the supporting face of the screw head.
- ✚ Save the files.

The bolt load acts on the support surface of the screw head.



Screws can be modeled in many different ways in FEM. We have selected a very simple method here.

This type of bolt load does not consider the bolt load increase or decrease that may occur due to operating force. If this is required, the screw must be generated via *1D BEAM* element that receives the diameter of the screw, or by volume elements with the realistic

geometry dimensions of the screw. The bolt load must then be defined with the function *Bolt Pre-Load* .

#### 4.2.4.15 Request Output of Contact Pressure

In contact analysis thus contact forces are calculated along the way. These can be utilized to calculate the surface pressure in the contact. Incidentally, there is another interesting quantity which can be calculated with contact analysis, namely the gap size of the contact. If these additional results shall be displayed in the post-processor, this must be requested in the so-called output requests. To do this, follow these steps:

- ✚ From the simulation navigator using RMB on node *Solution 1*, select the **EDIT** function.
- ✚ Go to the register *Case Control*.
- ✚ Select at *Output Requests* the function **EDIT**.
- ✚ In the following menu, go to the register *Contact Result*.
- ✚ Activate the switch *Enable BCRESULT Request*.
- ✚ Select at *Separation Distance* the option *SEPDIS*.
- ✚ Confirm twice with **OK**.

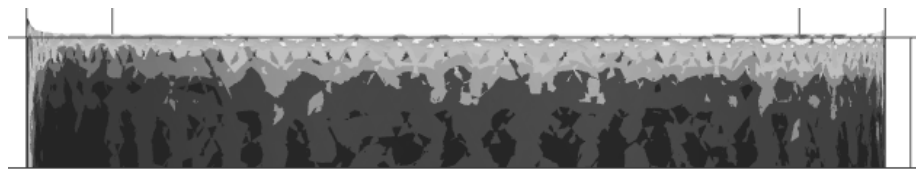
Additional results which are indeed calculated, but normally not output can be requested.

#### 4.2.4.16 Solve Solutions and Evaluate Results

The solutions are calculated in a conventional manner, whereby, due to the non-linearity, higher computation time (about two to five minutes, depending on the hardware) are expected.

- ✚ Run the function *Solve*.
- ✚ Change after the completion of the solution into the post-processor.
- ✚ For the contact pressure, set the result to *Contact Pressure – Nodal*.

In case that no results are calculated you should look for error messages in the *f06* file (FATAL). Here the solution progress is documented. After successful solution the deformation should be initially displayed and checked for plausibility.



The result of contact pressure with tetrahedral meshes trembles a little. This can be improved through better meshings in the second part of this task.

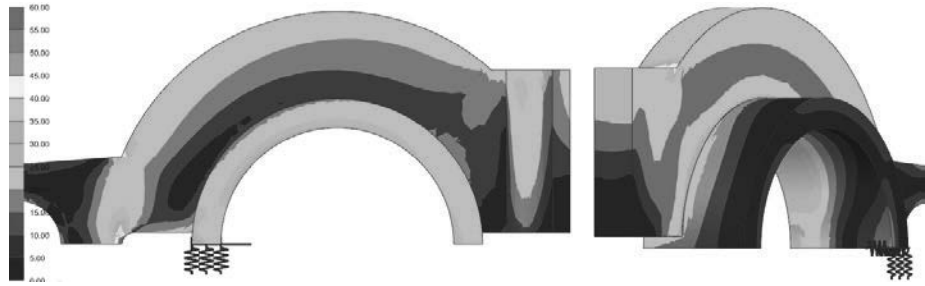
A colored version of this figure is available at [www.drbinde.de/index.php/en/203](http://www.drbinde.de/index.php/en/203)

The result for the contact pressure shows the expected pattern. Due to the unaligned nodes on the two sides of the contact the pattern is slightly non-uniform. But it certainly allows the use of values. When peaks are excluded at the very border, maximum values of about  $35 \text{ N/mm}^2$  yield.

- ✚ Set the result to von Mises stresses.



The von Mises stresses.



A colored version of this figure is available at [www.drbinde.de/index.php/en/203](http://www.drbinde.de/index.php/en/203)

The pattern shown in the figure above displays stress results. In the area of contact, the von Mises stress is similar to the result of the contact pressure previously shown. This has to be like that, because pressure and stress is the same thing. The contact pressure is just a more detailed analysis of stress in the contact area.

For this sample problem it means that the surface pressure is far below the permissible values and the tightening force is therefore acceptable.

#### 4.2.4.17 Part 2: Alternative Meshing with Hex-Tet Transition

The previous analysis with pure tetrahedral meshes of part 1 of this task has worked reasonably simple. Therefore, there is not much to suggest now to create even a more complex mesh. Nevertheless, in this second part an alternative meshing will be shown, which leads to more accurate and more uniform results in the contact area. Especially if fairly accurate results are desired in the contact region, there are two requirements for the meshing:

1. Hexahedral elements without middle node should be used.
2. The nodes of the two contact surfaces should be aligned with each other.

To meet the requirements for a very high quality mesh, some preparations need to be done.

If we want to fulfill these two requirements, we have some work to do. Elements without mid nodes are either eight node hexahedrons (Hex8) or four node tetrahedra (Tet4). Tet4 elements should be avoided if possible, because of their significant inaccuracy. Therefore, only the eight node hexahedron elements come into question, which are often used because of their high uniformity and good accuracy.

With good accuracy, we mean here that even in cases of coarse meshes accurate results appear. The Tet4 elements are of course accurately, but only at very, very fine meshes. However, hexahedral elements have the characteristic that they can be created just on extrusion capable geometry. To use hexahedrons, the model needs to be decomposed into parts such that each part is extrudable for itself.

However, this method would be feasible for our model, but in general much more complex than the simple tetrahedron meshing. A good way is to work with transitions. We can therefore mesh only the area of contact with hexahedra, then go to tetrahedron and use pyramidal elements in the transition area for the correct coupling of the two element types. This method is often very suitable, because it is very flexible, and so we use this route.

# Index

2D Contact 19  
3D contact 63  
64-Bit 10

## A

Adams 10  
ADINA 152  
Ampere's law 300  
analytical solution 372  
Animation 20  
approval processes 354  
Articulation 15, 31  
automatic time stepping 265  
AUTOMPC 182  
axisymmetric 311

## B

beam 187  
beam element 187–190  
beam theory 171  
bolt load 220  
boundary conditions 116  
boundary layer resolution 282  
Bushing 19

## C

Cable 19  
CAEAnalysis 353, 364  
CAEGeometry 353  
CAEManager 364–368  
CAEModel 353, 364  
cams 12  
capacitance 299  
capacitor 299  
CFD 271  
CGAP 209  
checking element shapes 128  
clamping element 207  
clamping seat 207

clamping situation 14  
clearance 14  
cloning 200  
coil 299, 301, 303, 311, 314, 317–319, 321, 324,  
326–329, 331, 334  
coil spring 185  
collision check 58  
collisions 16  
combination 176  
Component-based Simulation 25  
conduction losses 318, 331, 348–349  
conflict situations 24  
connection 166  
conservation equations 272  
Constant driver 31  
Constant Velocity 18  
Constraints 116  
contact 249  
contact non-linearity 84  
convection boundary condition 146  
convergence 135  
convergence control 291  
convergence criterion 275, 276  
convergence validation 136  
Co-Simulation 17, 24  
coupled systems 150  
coupling elements 177  
Create Sequence 20  
Curve on Curve 20  
cylindrical joint 18, 43

## D

damper 12, 19  
damping 64, 206  
data model 354, 363  
data protection 352  
default settings 10, 66  
determined degrees of freedom 15  
dielectric permeability 302  
dielectric relationship 302  
Direct Matrix Abstraction Programming 151

displacement function 371  
 displacement results 120  
 DMAP 151  
 DMU 3–4  
 driver 17, 30  
 dynamics 24, 59

## E

Eddy Current losses 318, 331, 348–349  
 Eddy Currents 304  
 edge subdivisions 233  
 eigenfrequencies 87, 199  
 electrical conductivity 302  
 electrical engineering components 297  
 electrical field theory 297  
 electrodynamics 299, 305–306  
 electrokinetics 299, 305–306  
 electromagnetic field analysis 150  
 electrostatics 299, 305–306  
 enforced displacement 194  
 Environment 17, 24  
 equilibrium condition 372  
 equivalent stress hypothesis 123  
 evaluation of accuracy 91  
 excitation 206

## F

f06 file 263  
 Faraday's law 301  
 ferromagnetic 302  
 finite-volume method 272  
 fixed joint 18, 22, 63  
 Flexible Body Dynamics 17, 25  
 Flexible Link 19  
 flow analysis 272  
 flow boundary conditions 284  
 flow surfaces 284  
 Force 20  
 forced movement 338, 340  
 four-node tetrahedrons 127  
 friction 64  
 frictionless sliding 289  
 full wave 299, 305, 307  
 Full Wave (High Frequency) 299, 307  
 Function driver 31  
 Function Manager 19  
 FVM 272

## G

gap elements 152  
 gear 19, 31  
 General Motion 339  
 geometric nonlinear analysis 230  
 Graphing 20, 45

## H

Harmonic driver 31  
 heat flux 147  
 heat transfer 89  
 hexahedral elements 127  
 HEX Solid Meshing 252  
 high frequency 299, 307  
 Hooke's law 83, 372

## I

induction law 301  
 inductivity 299, 326  
 inlet 286  
 Inline 18  
 installation 308–310  
 Interference 20  
 iron losses 348–349

## J

joint primitives 18

## L

large deformation 229, 249  
 large displacement 85, 229, 249, 261  
 leaf spring 229  
 learning tasks 5  
 library 61  
 licensing 308  
 lift force 293  
 lifting off contacts 60  
 linear buckling 90  
 linear statics 81  
 link 17, 25, 62  
 linked phase voltage 347  
 load transfer to FEM 20  
 load types 112  
 local mesh refinement 131  
 losses 318, 331, 348, 350

## M

machine portals 187  
 magnetic permeability 302  
 magnetic relationship 302  
 MAGNETICS 150  
 magnetodynamics 297–299, 305, 307  
 magnetostatics 297, 299, 303, 305, 307  
 Marker 18  
 mass properties 25  
 master model concept 21  
 Master Model Dimension 19  
 material equations 302  
 material law 372  
 material properties 109, 143, 283  
 MATLAB Simulink 17

- matrix form 375
- maturity tracking 352
- maximum distortion energy hypothesis 123
- maximum principal stress 123
- maximum tensile stress 196
- Maxwell's equations 298
- MBD program 12
- Measure 20
- memory 10
- mesh connections 144
- mesh fineness 85
- Mesh Mating Condition 144, 208
- Mesh Point 189
- middle node elements 127
- midsurface 161
- Motion Connections 28
- motion-driven systems 15
- Motion Joint Wizard 22
- Motor Driver 24
- motor libraries 17
- Moving Band 332, 339 – 340
- Multibody Dynamics 12
- multi-processor 10

## N

- named references 353, 362
- Newton's Method 261, 344
- non-linear contact 208 – 209
- non-linear effects 83
- non-linear geometry 261
- non-linear material 84
- non-linear stress-strain behavior 256
- notch factor 126
- notch stress 91
- NX Response Analysis 151
- NX/Thermal 150

## O

- ohm resistance 299, 303, 331, 348 – 349
- Ohm's law 302
- Opel RAK2 5
- opening 285
- Orientation 18
- outlet opening 287
- over-determinations 24
- over-determined degrees of freedom 24

## P

- Parallel 18
- parameterization 186
- PDM 3, 5
- perfect insulation 146
- Phase Shift 342
- phase voltage 343, 347
- piece of cake 142

- pivotable constraint 113
- planar joint 18
- Plant Input 20
- Plant Output 20
- plastic deformation 240
- plasticity 241
- PMDC-Motor 20
- point mass 201
- Point on Curve 19
- Point on Surface 20
- Poisson's Ratio 110
- polygon body 98
- polygon geometry 133, 233
- Populate Spreadsheet 20
- post-processor 118
- presetting 10
- press fit 207
- pressure distribution 293
- pretensioned bearings 186
- principle of linear FEM 82
- principle of the minimum of the potential energy 374
- principles of electromagnetic analysis 298
- processor 10
- process orientation 352

## R

- Rack and Pinion 19
- reaction force 186, 195
- RecurDyn 10
- redundant degrees of freedom 24, 41
- release and change processes 352
- residual tolerance 344
- resistance 299, 303
- restrictions of MBD 14
- revising 365, 367
- revisions 353, 362, 365 – 368
- revolute joint 18, 26
- ring-element-based method 141
- rivet joints 176
- rotational degrees of freedom 156, 179
- rotational driver 13

## S

- screw 18
- Sensor 18
- shape function 371, 373 – 376
- sheet 162
- shell elements 157
- signal chart 20, 24
- simulation data 351, 369
- simulation data management 352
- Simulation File View 97
- singularities 86, 137
- skin depth 304
- slider 18, 56
- Smart Point 18

- snap hook 249
- soft spring bearing 217–218
- Sol 101 151
- Sol 103 151
- Sol 106 152
- Sol 601 152
- space discretization 373
- spherical joint 18, 55
- spring 12, 19
- standard meshing 108
- starting behavior 340
- Steinmetz formula 303
- stiffness matrix 375
- Stitch Edge 162
- stress-strain behavior 372
- structural mechanics 14
- super elements 202
- surface roughness 284
- surface subdivisions 101
- Surface to Surface Contact 209
- Surface to Surface Gluing 208
- symmetry 141
- synchronization of the processes 352
- system of differential equations 12

## T

- TC\_CAE\_Defining 353, 362
- TC\_CAE\_Source 353, 362
- TC\_CAE\_Target 353, 362
- Teamcenter 352–355, 357–359, 361, 363–366, 368–369
- temperature boundary condition 145
- temperature field 139
- temperature gradient 147
- ten-node tetrahedral elements 127
- thermodynamic problem 150
- time-dependent travel path 258

- time step 291
- time step size 275
- TMG 150
- tolerances 42
- toolbar 16
- top-down method 35
- Torque 20
- Trace 20
- transport equations 272
- traverse path 258
- truss theory 372
- turbulence model 276

## U

- underdetermined 28
- undetermined degrees of freedom 15
- universal joint 18

## V

- vents 285
- version levels 352
- virtual product development 3–5
- von Mises 123

## W

- wake space 296
- wall thickness 165
- weak springs 216
- Whitney elements 314
- without redundancy 24

## Y

- Young's Modulus 110