

SIEMENS

Engineering Analysis with NX Advanced Simulation

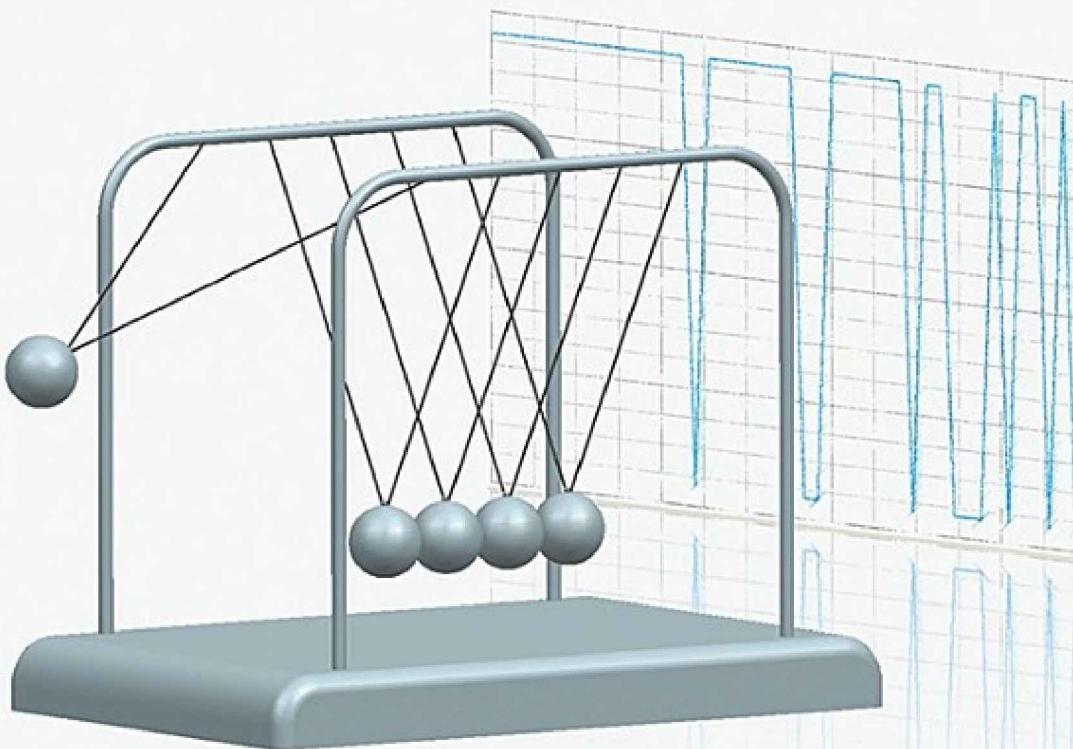
Pavel Goncharov

Igor Artamonov

Timur Khalitov

Sergey Denisikhin

Dina Sotnik



Engineering Analysis with NX Advanced Simulation

P. Goncharov, I. Artamonov, T. Khalitov

Copyright © 2014 P. Goncharov, I. Artamonov, T. Khalitov.
Siemens Product Lifecycle Management Software Inc.

All rights reserved. No part of this book may be reproduced, stored, or transmitted by any means—whether auditory, graphic, mechanical, or electronic—with written permission of both publisher and author, except in the case of brief excerpts used in critical articles and reviews. Unauthorized reproduction of any part of this work is illegal and is punishable by law.

ISBN: 978-1-4834-1731-8 (sc)
ISBN: 978-1-4834-1732-5 (e)

Library of Congress Control Number: 2014915839

The views expressed in this work are solely those of the author and do not necessarily reflect the views of the publisher, and the publisher hereby disclaims any responsibility for them.

Lulu Publishing Services rev. date: 11/11/2014

Engineering Analysis with NX Advanced Simulation

P. Goncharov, I. Artamonov, T. Khalitov

S. Denisikhin, D. Sotnik

*Dedicated to Steffen Buchwald –
teacher, partner and loved one*

Every book on CAE topic is a titanic effort. It is required to explain thousands of functions, hundreds of tools and dozens of use cases where one could utilize those tools. It is a known fact that when you have got a very powerful tool but no step by step manual, then the tool comes useless. We are sure that this book will inspire many professionals like researchers, stress analysts, aerodynamics specialists etc.. to get used to the broadest range of capabilities of NX Advanced Simulation environment.

– L.M.Khazin,

A.G. Yashutin,

Stress Department, Design Bureau “IRKUT Corporation”

The book provides an overview of NX capabilities in the area of engineering analysis. The main advantage of the book is that authors were able to detail and consistently describe the process of complex physical phenomena investigation within one book: starting from the choice of mathematical description and computational model preparation to results post processing and analysis. Brief description of mathematical models of physical phenomena, and description of numerical methods will allow users to competently use NX Advanced Simulation in their daily tasks. Use cases and examples from different physics come an efficient complementary to the main tool description. This book will be of interest to students, graduates and professionals in engineering analysis.

– A.Y. Slyunyaev,

PhD in Physical and Mathematical Sciences,

Head of IT Department,

JSC “Novosibirsk Aircraft

Production Association by VP Chkalov”

NX Advanced Simulation is a powerful environment to solve a wide range of CAE problems. Before the publishing of this book the process of NX Advanced Simulation learning was complicated due to lack of specialized books, which could facilitate the first steps of exploring the CAE system. The book materials enable professionals to independently learn capabilities of NX Advanced Simulation and begin to apply it in practice, bringing the benefit to enterprises they are working for. Separately, it should be noted that this book, unlike the other similar publications, contains the methodological approaches used during simulation process in addition to

technical capabilities. That is a very important part to educate a good CAE specialist.

– D.V. Shevchenko,
Expert researcher,
Siemens Corporate Technology

The book is quite detailed coverage of engineering analysis with NX Advanced Simulation. The book consists of two parts. The first part deals with general issues, tools, brief theoretical information on the engineering analysis in NX Nastran. There are a lot of examples, GUI snapshots, methods description for meshing, boundary conditions and result analysis. The second part represents various types of analysis – stress, modal, thermal, buckling, etc. It is worth noting the clarity of book material, as well as numerous practical examples of NX Advanced Simulation application. This one is the first edition of such high quality in Russia, dedicated to modern numerical methods for engineering analysis.

– M. Yu. Eltsov,
*Professor of the department of mechanical equipment
BSTU by V.G. Shukhov*

The book provides guidelines and detailed description of the approach to the creation of mathematical models based on CAD data (geometry obtained from designer) for solid parts and sheet metal parts. Undoubtedly the book will be interesting both to young engineers and experienced ones. The book represents all aspects of the design and simulation of structures from simple parts subject to standard loads, to complex assemblies with multi-disciplinary analysis. The book sets out most modern techniques used by highly skilled engineers worldwide to analyze structure's behavior and evaluate its performance in the shortest possible time.

– A.A. Mikhailov,
*Head of the Licensed Software
Research Institute of Materials and Technology SPbSPU*

TABLE OF CONTENTS

PART 1

Chapter 1. Getting Started with NX Advanced Simulation

- 1.1. The Procedure of Engineering Analysis in NX Advanced Simulation
- 1.2. Capabilities of NX Advanced Simulation
- 1.3. Calculations Types in NX Advanced Simulation
- 1.4. Simulation Model Structure and Creation Steps
- 1.5. Principles of the Finite Element Method

Chapter 2. Preparing a geometry model

- 2.1. Integration of design and calculations
- 2.2. Model Preparation toolbar commands
- 2.3. Midsurface toolbar commands
- 2.4. Commands of the Synchronous Modeling toolbar

Chapter 3. Creating and manipulating finite-element models

- 3.1. Structure of the FE model. Simulation Navigator
- 3.2. Collector fundamentals. Materials
- 3.3. Creating a finite-element model
- 3.4. Associated data of finite elements
- 3.5. Additional features for FE models
- 3.7. FE models of assemblies
- 3.8. Example. Creating a finite-element model of a U-shaped frame

Chapter 4. Creating a simulation model

- 4.1. Structure of the simulation model. Simulation Navigator
- 4.2. Coordinate systems and data fields
- 4.3. Loads, constraints on degrees of freedom, and simulation objects
- 4.4. Preparing the solution
- 4.5. Example. Creating and solving a simulation model

Chapter 5. Results visualization and post processing tools

- 5.1. Post Processing Navigator
- 5.2. Basic display features
- 5.3. Additional display features
- 5.4. Numerical output
- 5.5. Working with graphs
- 5.6. Additional result analysis tools

Chapter 6. Modelling composite structures

- 6.1. Special aspects of working with composite structures in NX
- 6.2. Creating composites based on physical properties
- 6.3. Creating composites using the global layup

- 6.4. Material types and microstructured materials
- 6.5. Solution parameters and viewing results
- 6.6. Summary of composite theory
- 6.7. Example. Static analysis of a composite product

PART 2

Chapter 1. Linear static analysis

- 1.1. Linear static analysis
- 1.2. Methods for solving systems of equilibrium equations
- 1.3. Solution types for linear static analysis
- 1.4. Setting solution parameters
- 1.5. Linear contact interaction
- 1.6. Gluing connections
- 1.7. Thermo elastic analysis
- 1.8. Optimization analysis
- 1.9. Example. Solving a problem with linear contact interaction

Chapter 2. Buckling analysis

- 2.1. Linear buckling analysis
- 2.2. Nonlinear buckling analysis

Chapter 3. Dynamic analysis

- 3.1. Fundamentals of dynamic analysis of structures
- 3.2. Accounting for inertial and elastic damping properties
- 3.3. Determining natural frequencies and free vibration modes of structures
- 3.4. Frequency response analysis
- 3.5. Transient analysis
- 3.6. Extra functionality for solving dynamic problems

Chapter 4. Nonlinear static and dynamic analysis

- 4.1. Introduction to nonlinear analysis. Special aspects of FE modelling
- 4.2. Geometrical nonlinearity
- 4.3. Nonlinearity of materials
- 4.4. Contact interaction
- 4.5. Solving nonlinear problems
- 4.6. Example. Analysis of thin-sheet stamping

Chapter 5. Analysis of heat and mass transfer

- 5.1. Capabilities of the heat transfer analysis module
- 5.2. Operating principle
- 5.3. Tools for specifying boundary conditions
- 5.4. Solution examples
- 5.5. Accounting for mass transfer in thermal problems

Chapter 6. Modelling fluid dynamics processes

- 6.1. General capabilities of NX Flow and NX Advanced Flow**
- 6.2. Constructing a fluid domain using surface wrap technology**
- 6.3. Using the Fluid Domain technique to build meshes**
- 6.4. Solution settings**
- 6.5. Specifying boundary conditions**
- 6.6. Modeling pressure losses**
- 6.7. Turbulence models**
- 6.8. Parallel computing in NX Advanced Flow**
- 6.9. Example solutions of problems**

PREFACE

NX™ Advanced Simulation is a feature-rich software system for multiphysics calculations. It can be useful for engineers in many areas of specialization – strength and dynamics studies, analysis of aerodynamic performance, internal and external flow of liquids and gases, cooling systems analysis, and experimental engineering. This book should be treated as a practical guide based on examples and problems that can be useful in everyday practice for domain experts as well as engineers who don't use NX Advanced Simulation in day-to-day activities.

The application of numerical methods to designing various structures and machines is driven by the necessity to keep improving the quality and reliability of products, and by opportunities to use novel structural materials, considering the harsh operation conditions contemporary products are exposed to. The effect of numerical engineering analysis technologies (CAE, Computer-Aided Engineering) is maximized if they are employed at early stages of design process. By that the cost of the product, the probability of malfunction, and the time to market can all be reduced. The behaviour of structures can also be studied by physical experiments with prototypes. This method lets the engineer evaluate the behaviour of the structure under different external influences. It is however costly, protracted and sometimes even completely inapplicable. Today the leading companies worldwide develop advanced competitive products using finite-element method (FEM) simulation to partially replace the expensive full-scale physical experiment with the cheaper and more expedient computational experiment. This is not surprising because the state of the art of computer and software technology allows solving difficult problems on powerful workstations and clusters relatively quickly. It is also important to note that full-scale experiments typically yield data for tens or hundreds of points. With numerical modelling the number of such points can be increased to hundreds of thousands, or, if necessary, more.

The target audience of this book are design, structural, and computing engineers, who are discovering the numerical analysis system, as well as those who want to expand NX Advanced Simulation knowledge and skills.

The book consists of two parts. In the first part we shall discuss basic tools for preparing the simulation model and analysing the simulation results. The second part deals with particular aspects of some types of engineering analyses in greater detail with practical examples. For an introduction to the NX Advanced Simulation system, both parts of the book can be recommended. For developing skills in solving particular applied problems, the second part is more pertinent.

The first chapter describes basic approaches to the NX Advanced Simulation, describes the structure of the simulation model, and suggests a calculation workflow. The primary types of calculations available in the NX Advanced Simulation are also described, as well as the principles of finite element method.

The second chapter deals with the most important commands and techniques for simplifying and modifying the initial geometry model to create a high-quality simulation model. Synchronous modeling commands are described in detail inasmuch as they are applicable to simulation models, together with other idealization commands.

The third chapter describes commands and recommendations for preparing finite-element models using different types of finite elements. Polygon geometry editing tools are discussed in detail, along with basic methods for creating a finite-element mesh, operations on elements and nodes, and techniques for preparing finite-element models of assemblies. The chapter also deals with commands used to assign physical properties to structures, and to create and store material properties in the library.

Boundary conditions with restrictions on degrees of freedom, as well as methods of applying such loads are presented in the fourth chapter.

The fifth chapter describes the presentation of results in the postprocessor. It discusses contour plots as well as aspects of creating plots and animated charts, and presenting the calculation data in tabular form.

The sixth chapter of the first part deals with special aspects of creating numerical models of parts made of composite materials. The chapter highlights the aspects one needs to take into account when simulating composite structures while developing the simulation model and while

processing the results.

The most important types of structure calculations are described in the second part of the book. The first chapter describes the analysis of structures' behaviour according to the theory of elasticity, and its parameters. It also describes optimization analysis. Analysis of linear and non-linear buckling is the subject of the second chapter. Basics of dynamical analysis as such and in NX Advanced Simulation are presented in the third chapter. The fourth chapter provides an overview of the most important types of nonlinearities and of the special considerations that need to be taken into account when solving static and dynamic problems. The fifth chapter describes formulating and solving the problem of complex heat exchange. Problems of computational fluid dynamics with formulations and solutions are described in the sixth chapter of the second part.

NX 8.0 and 8.5 were used to perform exercises in this book, however one can use NX 7.5 and NX 9 (with classic toolbars) to do most of the tasks similar way as described in this book. CAE and FEM models used for the exercises can be found here <http://www.siemens.com/plm/nxcaebook>

Historical roots of the NX Advanced Simulation

The primary solver in NX Advanced Simulation is the time-proven finite-element solver Nastran. Nastran is one of the first computation systems in the world. Nastran (NASA STRuctural ANalysis) was started in the middle of the previous century (1965), and the first commercial version of the solver code was released in 1972. In 2003 the source code as well as all intellectual property related to Nastran were bought by the company that is now Siemens PLM Software. Therefore, from 2003 onwards NX Nastran entered the marketplace. Since then Siemens PLM Software has significantly improved and extended NX Nastran.

NX Advanced Simulation in its current form also has benefited from a rich pedigree of other numerical analysis solutions. These were I-deas™ Master FEM, I-deas™ Laminate Composites, and I-deas™ Advanced Durability solutions developed by SDRC since 1967 and which are now property of Siemens PLM Software. Heat and mass transfer analysis modules that had been developed by Maya Heat Transfer Technologies since 1983, are now included into Siemens PLM Software's solution portfolio as NX

Flow/Advanced Flow, NX Thermal/Advanced Thermal, NX Space System Thermal, NX Electronic System Cooling. Adina technologies became the foundation for NX Nastran Advanced Nonlinear – the module for analyzing complex static and dynamic non-linear processes. Adina R&D company was founded in 1986 by Massachusetts Institute of Technology (MIT) professor Dr. K. J. Bathe.

Therefore, the CAE solution portfolio of Siemens PLM Software unites a large number of state-of-the-art, best-in-class technologies, such as Nastran, Adina R&D, I-deas™ CAE, Maya HTT, Recurdyne and others. From all the capabilities of these technologies Siemens PLM Software selected the ones best suited for each class of problems, uniting them in a single solution, which is actively developed and improved to allow analysing applied problems of highest complexity.

PART

1

Chapter 1. Getting Started with NX Advanced Simulation

This chapter describes the basic techniques and steps of numerical engineering analysis using the NX Advanced Simulation CAE system. This chapter answers the first practical questions that arise while working with the application. The user will also become acquainted with the finite element method for problems of dynamics and strength of machines and structures.

1.1. The Procedure of Engineering Analysis in NX Advanced Simulation

The structural engineer typically prepares a preliminary strength/performance evaluation of a structure using engineering methods based on representing the structure as simplified parts, whose stress-strain behaviour can be evaluated analytically. These evaluations might include using simple formulae to determine tensile, flexural, or torsional stress in beams, finding the deformation moments of inertia, reaction forces, etc. The structural engineer has to sift through a large volume of specialized literature to find the necessary expressions and laws. These approaches are substantially limited if applied to real complex structures, so they are being phased out in modern high-tech manufacturing and design. Numerical analysis systems enable the engineer to model structures and machines of any complexity with rational level of detail. The engineer gains a tool for analysing the actual stress and strain distribution in the structure. NX Advanced Simulation, based on the NX Nastran industrial solver (and other Siemens PLM Software solvers), lets the engineer work with different applications while staying within the unified NX design environment. The scalability of the NX Advanced Simulation module allows solving simple as well as the hardest problems from different domains of deformable solid mechanics, fluid mechanics, heat transfer, etc.

The primary steps of engineering analysis using the finite element method (Fig. 1.1) are:

- Creating an idealized *i-part* model. This step corresponds to going from the actual physical model representation to a modified

(simplified) mathematical model. Obviously, mathematical models have infinite degrees of freedom, therefore the problem for a complex model is not solvable in practice.

- Creating a discrete *FEM* model. This corresponds to limiting the degrees of freedom, i.e. discretization of the idealized model.
- Solving the system of resolvent equations corresponding to the chosen type of analysis.

One must keep in mind that numerical solution of the problem cannot be exact; each step of numerical modelling introduces a certain inaccuracy into the calculation result. To minimize the calculation error, the engineer must pay special attention to two steps: idealization and discretization. At the idealization step transition to the mathematical model is made, which can introduce a substantial or even drastic error into the result. At the discretization step the numerical solution must be checked for convergence. Convergence here is the tendency of the numerical solution result to converge to the correct one if the number of degrees of freedom increases.

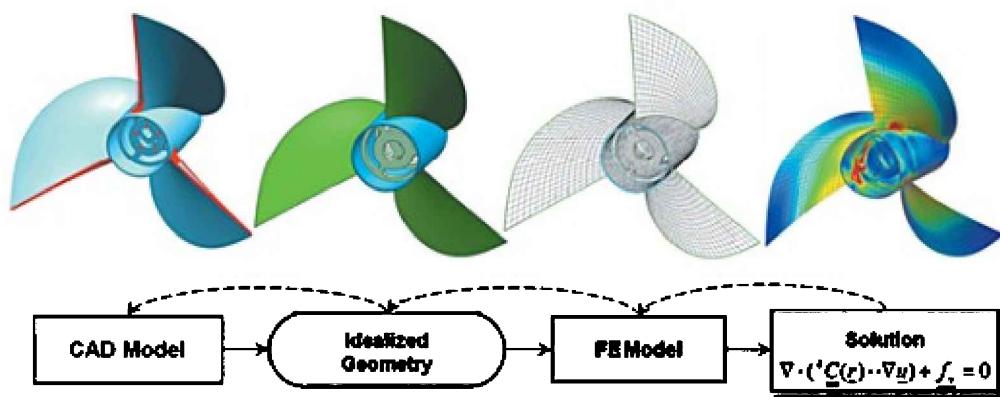


Figure 1.1. Engineering analysis workflow

Moreover, when doing numerical modelling, one must remember that FEM analysis always requires balancing the expertise of the engineer, the precision of results, available computational power, calculation time, and duration of simulation model development. The more detailed and well-discretized simulation models typically lead to a greater precision but require more time for computation as well as preparation. Conversely, the quality of the simulation model and the time spent on preparing it can directly reduce the calculation time. And expertise and knowledge contributed by the engineer is the most important determinant that can help balance the other

factors and achieve a practical solution of the problem at hand.

1.2. Capabilities of NX Advanced Simulation

NX Advanced Simulation is a multi-purpose module for finite-element modelling. It has a rich set of features for analysis modeling, simulating structural behavior and visualizing results. The module contains all tools that a CAE specialist needs, and supports a broad spectrum of engineering calculations. NX Advanced Simulation provides complete associativity of simulation models with CAD models, allowing rapid modification of the structure and the corresponding simulation model.

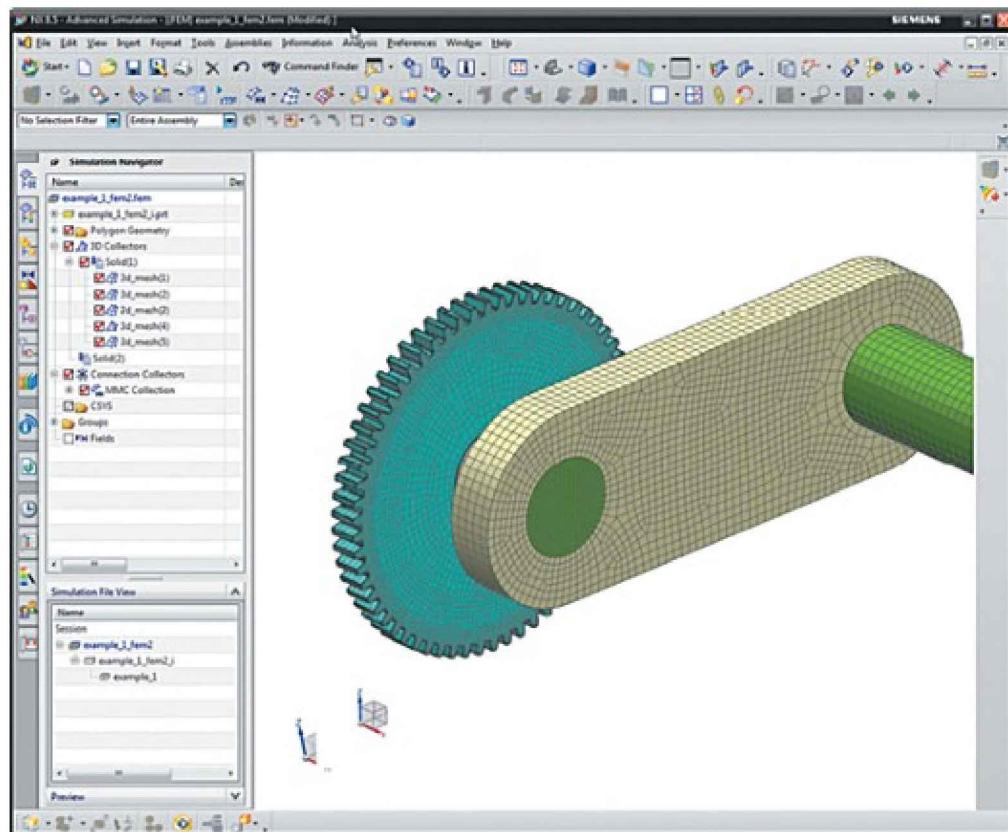


Figure 1.2. General view of NX Advanced Simulation

A distinguishing feature of NX Advanced Simulation is the ability to stay within the NX modelling environment while using various industrial solvers such as NX Nastran, MSC Nastran, ANSYS, LS-Dyna and ABAQUS. One needs only to specify the type of the required solver, and the system automatically represents all models, element types, properties, parameters,

matching conditions and solution options, using the terminology or “language” of the chosen solver and analysis type (Figure 1.3).



Figure 1.3. “Language” of the chosen solver

NX Advanced Simulation, one of the leading numerical engineering analysis systems, provides a set of tools and features for numerical analysis of any level of complexity – from the simplest estimation calculations to analysing processes of the greatest complexity (such as impact tests, process problems, correlated heat and mass transfer problems, etc). The structure of data and parameters of the simulation model is represented as a non-chronological tree, where all parameters can be accessed from the main menu, the model tree, or the graphics area of the screen. This intuitive structure ensures that users with any level of skill can start using the system in the shortest time possible.

Going from the physical model to the mathematical NX Advanced Simulation model, a set of specialized tools allows adapting the geometry of the CAD design for FEM analysis. For example, CAE engineers can simplify the model without consulting the designer by removing small geometry elements, blanking holes and blends, creating midsurfaces, and carrying out Boolean operations and body division operations to improve the quality of the calculation mesh.

One of the distinguishing features of mathematical simulation in NX Advanced Simulation is the logical separation of the simulation and FE models. Therefore, at each moment only one model is active, thereby making the job of the engineer much more orderly and cutting down on

machine resource use. This principle also allows engineers to carry out several different analysis types for a single FE model.

NX Advanced Simulation efficiently creates FE models with high-quality meshes and allows using the full spectrum of existing finite element types (0D, 1D, 2D, and 3D) thus being able to control the FEM decomposition to a great extent. To improve the FE model quality considering specific topology of the geometry, special tools exist that can remedy problematic areas of the geometry. One of the most important advantages of NX Advanced Simulation is the automatic tracking of changes to geometry and the FE model with automatic updating of the simulation model.

1.3. Calculations Types in NX Advanced Simulation

NX Advanced Simulation can analyse the structure from the point of view of different processes related to its functioning. To perform the calculation, the user can use: linear/non-linear strength analysis, structural dynamic behaviour analysis, analysis of the item affected by rapid non-linear processes, thermal analysis, fluid flow analysis, optimization analysis, correlation analysis, etc. When using the **NX Advanced Simulation** application to create a calculation model, the first thing to determine is the solver and the type of analysis (Fig. 1.4) to use for the model at hand. Depending on the chosen solver, the system automatically sets up the interface and the “language” of commands and functions of the pre- and postprocessor.

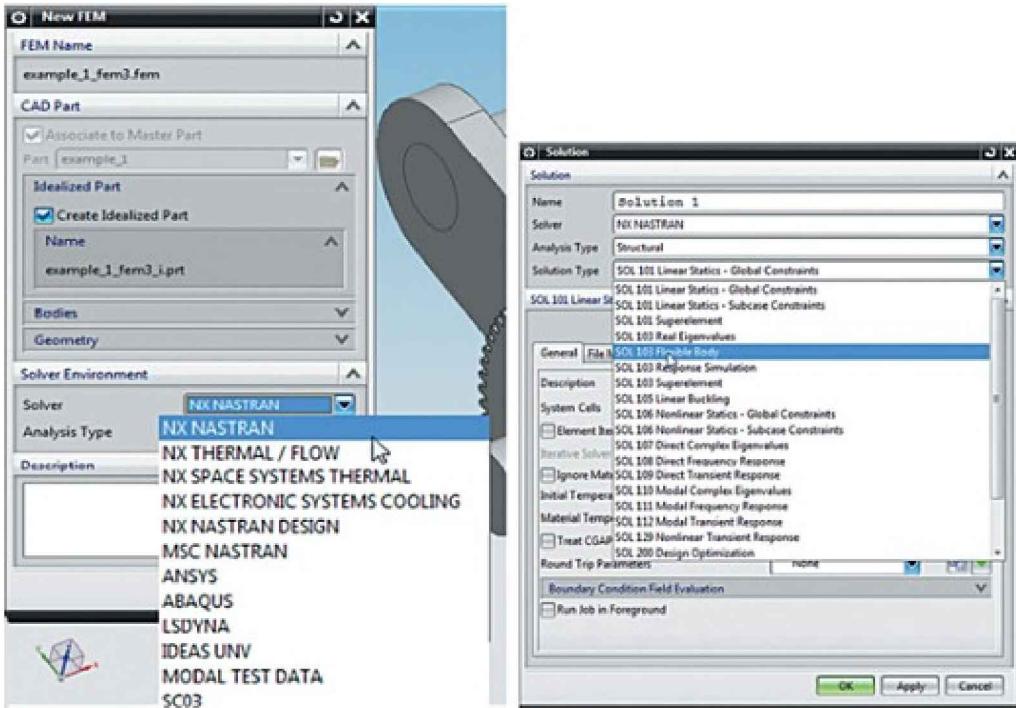


Figure 1.4. Types of solvers and analyses of the NX Nastran solver

The primary analysis types available in the NX Nastran solver are: linear static analysis (SOL 101), eigenfrequency and free vibration mode analysis (SOL 103), time and frequency response analysis (SOL 108, 109, 111, 112), structure buckling analysis (SOL 105), basic nonlinear analysis (SOL 106), transient process analysis (SOL 129), heat transfer analysis (SOL 153), nonlinear analysis based on implicit integration schemes (SOL 601), nonlinear dynamic analysis based on explicit integration schemes (SOL 701), optimization (SOL 200).

In addition, more solver type options are available:

- NX Thermal/Flow – these solvers can analyse heat transfer problems and perform computational fluid dynamics (CFD) analysis. These two solvers can be used independently or together to get results of thermal analysis, fluid dynamics analysis, and the correlated heat and mass transfer analysis.
- Thermal analysis of spacecraft (NX Space Systems Thermal) – this is an industry specific solution for thermal analysis of spacecraft and orbital/interorbital systems
- Thermal analysis of electronic systems (NX Electronic Systems

Cooling) – this is an industry specific solution for analysing cooling systems for electronic equipment. It includes heat transfer analysis and CFD for comprehensive analysis of heat removal systems.

- MSC NASTRAN, ANSYS, ABAQUS, LS-DYNA, IDEAS UNV – the simulation model can be written as an input file for the corresponding numerical analysis system.

1.4. Simulation Model Structure and Creation Steps

When performing any FEM simulation in NX Advanced Simulation one must have a clear understanding of the type of the problem to be solved and the type of results needed to evaluate the solution. If the user formulates the problem incorrectly, the solution will be imprecise or wrong.

The procedure of engineering simulation of structures using the finite element method in NX Advanced Simulation can be split into several stages (Figure 1.5). The process of model creation and calculation involves creating some files with certain types of data relevant to the simulation model. To use NX Advanced Simulation effectively, one must clearly discern which data are stored in which file, and which file needs to be active when creating the simulation model and working with it.

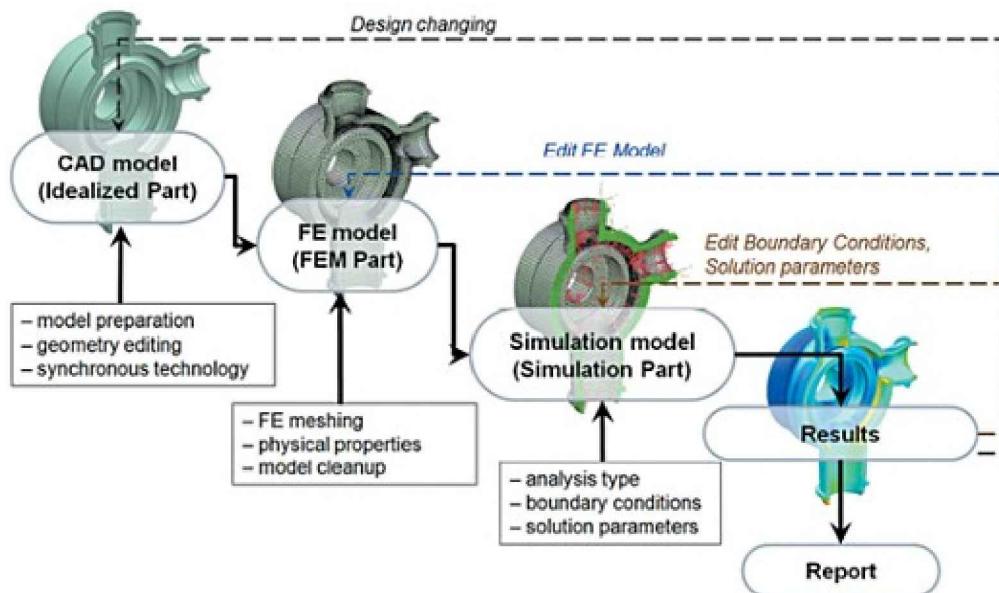


Figure 1.5. Stages involved in creating a simulation model

The stages of creating a simulation model are detailed below:

1. Creating an idealized geometry model (*/idealized Part*)

To perform FEA effectively, the geometry model must be accurate but as simple as possible. To achieve this, the entire source geometry (master model) must be subjected to idealization. All geometry elements, which increase the complexity of the simulation model (fabrication holes, blends, chamfers) but do not influence the expected results of the simulation, should be eliminated. If the source geometry includes visible surface defects, the corresponding geometry elements should be rebuilt. NX provides special tools to achieve this.

Idealization of the geometry (i.e. modification or simplification) does not entail modifying the master model. The system automatically creates an idealized geometry model with a corresponding “*name_fem_i.prt*” file. This file is created together with the *FEM* model file or the *S/M* simulation file.

2. Creating an FE model (*FEM Part*)

The “*name_fem.fem*” file is created for the FE model. In addition to the mesh itself, this file defines and stores physical properties of model parts, such as material properties, and parameters of shell and rod elements.

The geometry in the newly created *FEM* file is polygonal, which means it consists of facets, vertices, and edges due to the discrete representation. In subsequent stages, it is this polygonal geometry that is used to define specialized mesh generation rules, such as the number or the size of elements on a geometry object, or the geometry abstraction. The *FEM* file is associatively linked with the idealized geometry. This means it can be automatically updated whenever the source or the idealized geometry is modified.

NX Advanced Simulation application can perform numerical simulation for separate parts of structures as well as for several parts merged into an assembly. In case of working with an assembly the file structure of the FE model (*Assembly FEM* or *AFEM*) slightly differs from the structure of the general FEM. To create the assembly finite-element model, first the FE models of individual parts are created, and then the assembly FEM (*AFEM*) is created, where all existing component FE models are brought together as an assembly. The FE models of individual parts are automatically positioned in the assembly relative to each other in accordance with their initial positions in the CAD assembly. If the original CAD assembly is not available,

FE models of individual parts can be positioned manually using special tools.

3. Creating a simulation model (*Simulation Part*)

The *SIM* file contains information on the formulation of the problem. That means in this stage, boundary and initial conditions are defined for the developed FE model, as well as possible collision conditions, one or more analysis types, and solver options.

The created “*name_sim.sim*” simulation file contains all parameters and properties of the structure’s behaviour, simulation cases, solver settings such as solution type, solution step, simulation objects (collision boundary conditions, etc.), loads, limitations, and overridden physical properties. Several *SIM* simulation files can be created and associatively linked to a single *FEM* file.

4. Solving the problem numerically (*Solution*)

This stage does not require direct involvement of the engineer but is usually accompanied by the so called solution progress monitoring. To identify problems in the solution progress at an early stage, one should monitor convergence of the solver, convergence of the collision algorithm, and non-linear or unsteady solution history. If divergence of the solution or other difficulties is discovered, the calculation can be interrupted to adjust the *FEM* or *SIM* files.



Figure 1.6. Monitoring the solution progress

5. Analyzing the results (*Results*)

When correct results are obtained as an OP2 file (for NX Nastran solver), these results are analyzed, any necessary plots and distributions are created, and a report is composed.

The general structure of a typical simulation model in NX Advanced Simulation is shown in Figure 1.7.

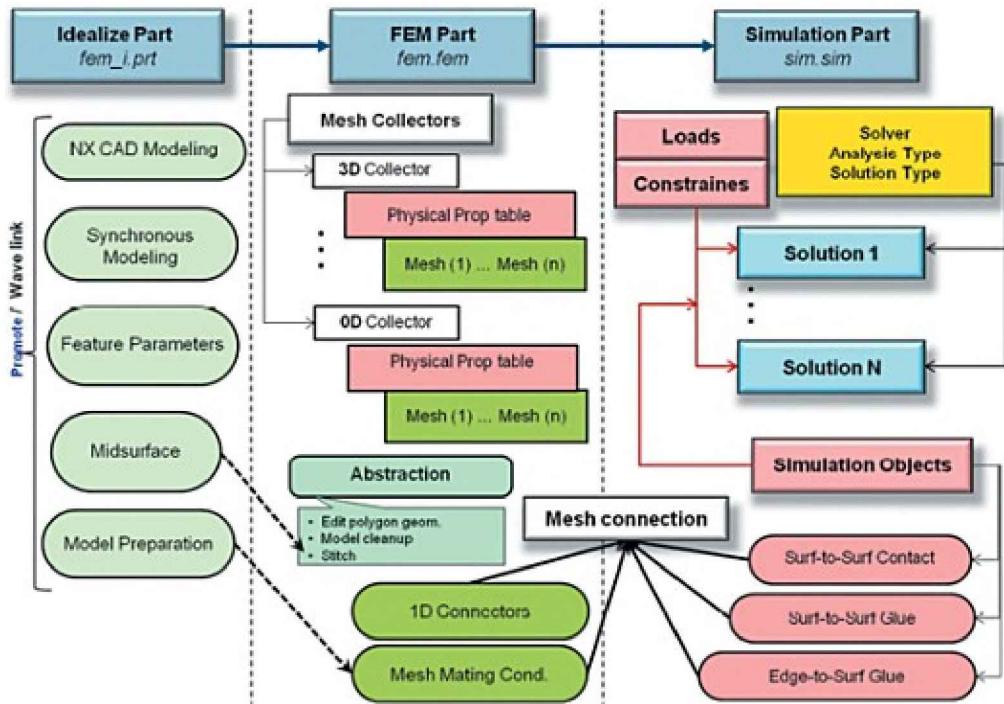


Figure 1.7. Diagram of a typical NX Advanced Simulation model

There are several advantages to manipulating CAE data in the NX Advanced Simulation model structure:

- Files with *.sim* and *.fem* extensions can be used by a PLM system to manage numerical modelling data and processes.
- Opening (loading) of master and idealized geometries is optional, therefore a smaller volume of RAM is used, and the system works faster.
- The intuitive logical structure makes it easier to work with highly complex models.
- Reuse of *FEM* simulation mesh files can significantly improve resource use efficiency.

NX Advanced Simulation application is a flexible configurable environment, which allows using different workflows to achieve a particular objective depending on the problem being solved and personal preferences. That being said, two principal workflows can achieve a correct result for most of simulation cases.

In the “explicit” workflow recommended for most models one first specifies

the material, physical properties and mesh properties in the FEM mesh collectors, and then generates the mesh itself. A FEM mesh collector is an element of the simulation model tree containing information on the type, properties and parameters of the simulation mesh. This workflow is useful for building complex models containing multiple bodies, materials and FEM meshes. It guarantees the transparency of the model's properties and decreases the risk of simulation or calculation errors.

Simple structures with a single solid or surface body and one material can be processed using the “automatic” workflow. This workflow allows quick and automatic creation of *FEM* and *SIM* simulation model files with all necessary collectors. In this case, object properties are inherited from the geometry model by default.

NX Advanced Simulation can create multiple *FEM* files for a single part. For example, a large (“coarse”) mesh and a more detailed mesh can be generated for the same geometry object. To associate a new *FEM* file with an existing idealized part, in the **New FEM** dialog select the *Associate to Master Part* checkbox and choose an idealized part from the list of open parts, or click *Open Part* and open an idealized part. To associate the new *FEM* file with a new idealized part, in the **New FEM** dialog select *Associate to Master Part*) checkbox and choose a master part from the list of open parts. In this case, to create a new idealized geometry select *Create Idealized Part*. Click **OK** to create a new idealized part based on the master model.

Much like multiple *FEM* models, one can create multiple *SIMs*, as well as multiple solutions in a single simulation file. To reuse boundary conditions when creating a simulation model with several solutions, drag the boundary condition to the new simulation case. By this way one can ensure that all simulation cases use the same material properties and physical properties of objects. If multiple *SIMs* are used, structure properties can be modified for each simulation case by overriding. By overriding properties in the *SIM* files one can modify properties of selected materials, physical properties, or element attributes without modifying the simulation mesh (the *FEM* file) accordingly. When the model containing overridden physical-mechanical properties is simulated, the system uses values from the *SIM* file level instead of original values. In this way many different materials can be

studied in the same model without spending extra time or cluttering the disk space with duplicate files. Property overwriting is also useful to quickly evaluate applicability of different element thicknesses when working with surface models.

1.5. Principles of the Finite Element Method

The finite element method (FEM) is a variation-differential method, which is based on representing an original area with a complex boundary as a collection of simple subareas (finite elements) [27]. Representing an area of interest as a collection of subareas means discretizing a continual problem, i.e. replacing an infinitely great number of degrees of freedom of the real body with approximately adequate but finite number of degrees of freedom. The subsequent derivation of equations for the collection of elements from variation principles of mechanics determines the variation character of the method.

The most important advantage of FEM is the ability to obtain arbitrarily precise solutions of any practical solid mechanics problem [2].

The quasi-static elastic problem in displacement for a heterogeneous anisotropic medium involves solving three differential equations of equilibrium relative to the components of the displacement vector:

$$\nabla \cdot (\mathbf{C}(\mathbf{r}) \cdot \nabla \mathbf{u}) + \mathbf{f}_v = 0$$

where $\mathbf{r} = x_k \mathbf{e}_k = x_1 \mathbf{e}_1 + x_2 \mathbf{e}_2 + x_3 \mathbf{e}_3$ – radius-vector of the point; $\mathbf{u} = u_k \mathbf{e}_k$ – displacement vector; $\nabla = \mathbf{e}_k \frac{\partial}{\partial x_k}$ – Hamiltonian operator; $\mathbf{C}(\mathbf{r}) = C_{ik} \mathbf{e}_i \mathbf{e}_k$ – elastic modulus tensor; $\mathbf{f} = f_k \mathbf{e}_k$ – vector of volume forces; the “.” symbol stands for convolution.

The system of equilibrium equations defining the behaviour of the solid in points of its volume V , is supplemented with conditions on its bounding surface S – kinematic, load or mixed boundary conditions.

$\mathbf{u} _S = \mathbf{u}_S(\mathbf{r})$
$\mathbf{n} \cdot (\mathbf{C} \cdot \nabla \mathbf{u}) _S = f_S$
$\mathbf{u} _{S1} = \mathbf{u}_S(\mathbf{r}); \mathbf{n} \cdot (\mathbf{C} \cdot \nabla \mathbf{u}) _{S2} = f_S; S = S1 \cup S2,$

where $\mathbf{u}_s = u_{s,k} \mathbf{e}_k$ – displacement vector defined at the boundary; $\mathbf{f}_s = f_{s,k} \mathbf{e}_k$ – surface load vector defined at the boundary; $\mathbf{n} = n_k \mathbf{e}_k$ – outward normal to the surface of the body. Combined boundary conditions are possible as well, when one or two of the three equations, which must be defined at each point of the surface S , are formulated in terms of displacement, and the others in terms of forces.

In case of small strain ($|\nabla u| \ll 1$), the tensor of strain $\boldsymbol{\varepsilon}$ is expressed in terms of the displacement vector using Cauchy equations $\boldsymbol{\varepsilon} = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) = (\nabla \mathbf{u})^S = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\mathbf{e}_i \mathbf{e}_j$, where $(\cdot)^T$ – transposition operation; $(\cdot)^S$ – symmetrizing operation.

It is necessary to find the vector function $\mathbf{u}(\mathbf{r})$ in a domain V bounded by surface S .

The basic concept of FEM is building discrete FE models of the domain and the continuous function. The V domain is approximated by a finite number of non-intersecting subdomains called finite elements, which have common nodes. The vector function $\mathbf{u}(\mathbf{r})$ is interpolated at each FE polynomial, which is defined by node values of the sought-for function $\mathbf{u}(\mathbf{r})$.

The V domain is represented as a collection of finite elements $\mathbf{V}^{(e)}$, $e = \overline{1, n_e}$; n_e – total number of FEs. The finite elements have common nodes, each of which has a number J , $J = \overline{1, n_p}$, n_p – the total number of nodes. The FE model of a domain is characterized by a global vector of node coordinates \mathbf{X}

$$\mathbf{X}^T = (\dots, X_J^T, \dots)_{J=\overline{1, n_p}}$$

The FE node coordinate vector $\mathbf{x}^{(e)}$ is obtained from the global vector \mathbf{X} using incidence matrix $a_g^{(e)}$ as follows:

$$\mathbf{x}^{(e)} = a_g^{(e)} \mathbf{X}$$

The choice of element type, shape, and number of nodes depends on the problem at hand and the necessary precision.

As the most important unknowns called degrees of freedom, FEM adopts node values of the sought-for displacement vector function and, if necessary, its derivatives.

The following vectors are introduced:

\mathbf{U} – global vector of nodal unknowns for the entire structure

$$\mathbf{U}^T = (\dots, U_j^T, \dots)_{j=1, n_p}$$

$\mathbf{u}^{(e)}$ – local (element) vector of nodal unknowns, which is formed from the global nodal displacement vector \mathbf{U} using the kinematic constraint matrix $a_f^{(e)}$ (incidence matrix) in the following way:

$$\mathbf{u}^{(e)} = a_f^{(e)} \mathbf{U}$$

After selecting the nodal unknowns, the interpolation polynomial is built, which expresses the law of variation of the sought function within the boundary of a FE through the values of its nodal unknowns.

The displacement vector $\mathbf{u}^T = (u_1, u_2, u_3)$ at an arbitrary FE point (e) with radius vector $\mathbf{x}^T = (x_1, x_2, x_3)$ is defined as

$$\mathbf{u}(\mathbf{x}) = N^{f(e)} \mathbf{u}^{(e)} = N^{f(e)} a_f^{(e)} \mathbf{U}$$

$$\text{where } \mathbf{x} = N^{g(e)} \mathbf{x}^{(e)} = N^{g(e)} a_g^{(e)} \mathbf{X}$$

Here $\mathbf{N}^{f(e)}$ – interpolation polynomial matrix; $\mathbf{N}^{g(e)}$ – approximating function (shape function) matrix.

To simplify formulating equations of the matrix form of the strain and stress tensor, it can be represented as a “strain vector” and “stress vector”. The strain vector is defined as

$$\boldsymbol{\varepsilon} = (\nabla \mathbf{u})^S \rightarrow \boldsymbol{\varepsilon} = \mathbf{B}^{(e)} \mathbf{u}^{(e)}$$

and the stress vector as

$$\boldsymbol{\sigma} = \mathbf{C} \cdot (\boldsymbol{\varepsilon} - \boldsymbol{\alpha} \Delta T) \rightarrow \boldsymbol{\sigma} = \mathbf{D}^{(e)} (\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_0) + \boldsymbol{\sigma}_0$$

where $\mathbf{B}^{(e)}$ – gradient matrix, $\mathbf{D}^{(e)}$ – elasticity matrix, $\boldsymbol{\varepsilon}_0$ – initial strain (thermal strain) vector, $\boldsymbol{\sigma}_0$ – initial stress vector.

The principle of virtual displacement is written as follows:

$$\int_V \delta \epsilon^T \cdot \sigma dV = \int_V \delta U^T \cdot f_v dV + \int_{S_1} \delta U^T \cdot f_s dS$$

where $f_v (f_s)$ – vector of volume (surface) forces.

After substituting the following relationships into the virtual displacement principle:

$$U = N^{f(\epsilon)} U^{(\epsilon)} = N^{f(\epsilon)} a_f^{(\epsilon)} U^{(\epsilon)}$$

$$\epsilon = B^{(\epsilon)} U^{(\epsilon)} = B^{(\epsilon)} a_f^{(\epsilon)} U$$

$$\sigma = D^{(\epsilon)} (\epsilon - \epsilon_0) + \sigma_0 = D^{(\epsilon)} (B^{(\epsilon)} a_f^{(\epsilon)} U - \epsilon_0) + \sigma_0$$

considering additivity

$$\int_V (\dots) dV = \sum_e \int_{V(e)} (\dots) dV$$

ends up with the following:

$$\begin{aligned} \delta U^T \left(\sum_e a_f^{(\epsilon)}{}^T \left(\int_{V(e)} B^{(\epsilon)}{}^T D^{(\epsilon)} B^{(\epsilon)} dV \right) a_f^{(\epsilon)} \right) U = \delta U^T \left(\sum_e a_f^{(\epsilon)}{}^T \left(\int_{V(e)} N^{f(\epsilon)}{}^T f_v dV + \int_{S_2(e)} N^{f(\epsilon)}{}^T f_s dS + \right. \right. \\ \left. \left. + \int_{F(e)} B^{(\epsilon)}{}^T D^{(\epsilon)} \epsilon_0 dV - \int_{F(e)} B^{(\epsilon)}{}^T \sigma_0 dV \right) \right) \end{aligned}$$

Due to the arbitrary nature of δU in the volume and the part of the surface S_2 , where displacements are not defined, we can write

$$KU = F, \quad (1.1)$$

where the following notation is used:

$$K = \sum_e a_f^{(\epsilon)}{}^T k^{(\epsilon)} a_f^{(\epsilon)} \quad \text{– global stiffness matrix,}$$

$$k^{(\epsilon)} = \int_{V(e)} B^{(\epsilon)}{}^T D^{(\epsilon)} B^{(\epsilon)} dV \quad \text{– FE stiffness matrix (e),}$$

$$F = \sum_e a_f^{(\epsilon)}{}^T f^{(\epsilon)} = \sum_e a_f^{(\epsilon)}{}^T (f_v^{(\epsilon)} + f_s^{(\epsilon)} + f_{\sigma 0}^{(\epsilon)} + f_{\sigma 0}^{(\epsilon)}) \quad \text{– global vector of nodal forces.}$$

At the same time

$$\begin{aligned} \mathbf{f}_v^{(e)} &= \int_{V(e)} N^{f(e)^T} \mathbf{f}_v dV \\ \mathbf{f}_s^{(e)} &= \int_{S\Omega(e)} N^{f(e)^T} \mathbf{f}_s dS \\ \mathbf{f}_{\varepsilon_0}^{(e)} &= \int_{V(e)} B^{(e)^T} D^{(e)} \boldsymbol{\varepsilon}_0 dV \\ \mathbf{f}_{\sigma_0}^{(e)} &= - \int_{V(e)} B^{(e)^T} \boldsymbol{\sigma}_0 dV \end{aligned}$$

where $\mathbf{f}_v^{(e)}$, $\mathbf{f}_s^{(e)}$, $\mathbf{f}_{\varepsilon_0}^{(e)}$, $\mathbf{f}_{\sigma_0}^{(e)}$ – vectors of FE nodal forces (e), which are statically equivalent to the volume \mathbf{f}_v and surface \mathbf{f}_s forces acting on the body, as well as fields of initial strain $\boldsymbol{\varepsilon}_0$ and stress $\boldsymbol{\sigma}_0$.

Based on the calculated element stiffness matrices $k^{(e)} (e = \overline{1, n_e})$ and load vectors $\mathbf{f}^{(e)} (e = \overline{1, n_e})$ the kinematic constraint matrix $\mathbf{a}_f^{(e)}$ allows working out the global stiffness matrix of the entire structure \mathbf{K} (finite element ensemble) and the global external load vector \mathbf{F}

$$\begin{aligned} k^{(e)} (e = \overline{1, n_e}) \rightarrow \mathbf{K} &= \sum_{e=1}^{n_e} \mathbf{a}_f^{(e)^T} k^{(e)} \mathbf{a}_f^{(e)} \\ \mathbf{f}^{(e)} (e = \overline{1, n_e}) \rightarrow \mathbf{F} &= \sum_{e=1}^{n_e} \mathbf{a}_f^{(e)^T} \mathbf{f}^{(e)} \end{aligned}$$

This process is known as assembling.

After solving the system of linear algebraic equations (1.1) using interpolation processes at any point of the structure, one can calculate displacements and stresses

$$\begin{aligned} \mathbf{U} &= N^{f(e)} \mathbf{U}^{(e)} = N^{f(e)} \mathbf{a}_f^{(e)} \mathbf{U} \\ \boldsymbol{\varepsilon} &= B^{(e)} \mathbf{a}_f^{(e)} \mathbf{U} \\ \boldsymbol{\sigma} &= D^{(e)} (B^{(e)} \mathbf{a}_f^{(e)} \mathbf{U} - \boldsymbol{\varepsilon}_0) + \boldsymbol{\sigma}_0 \end{aligned}$$

Chapter 2. Preparing a geometry model

Studies show that CAE analysts spend most of their time on preparing simulation models. This time is spent mostly on building a mathematical model by adapting, simplifying, and modifying the source geometry that represents the real model of the object. This adaptation (also called idealization) is needed because the requirements that a simulating engineer has for geometry will differ from the designer engineer's requirements. Moreover, the analyst can develop certain mathematical hypotheses that make certain geometry modifications necessary.

2.1. Integration of design and calculations

CAE systems are typically separate from CAD applications. This means they cannot process geometry quickly and efficiently or apply the full spectrum of features for processing, editing, and extending an existing geometry model (when working with solids as well as surfaces and rod structures). If the existing geometry needs to be modified due to the results of the analysis, the engineer often has to discard the existing discrete FE model and recreate it after editing the geometry. Large-scale modification of geometry in CAE systems is difficult and involves modifying the original CAD system objects. The analyst therefore needs an extra skill set for working in a different environment.

Moreover, analysts and structural engineers need to interact with various people within an enterprise which makes coordination of simulation and design models particularly relevant. This coordination greatly amplifies the advantages of automated design, and elevates virtual engineering of products to a new level.

Engineers also encounter situations where they need to numerically analyze a product represented in neutral geometry interchange formats (JT, Parasolid, IGES, STEP, and others), or use drawings as source data. In this case the availability of tools that allow to modify and build 3D models is also a necessary component of a numerical engineering analysis software package. Tools for creating geometry models of fluid domains are particularly indispensable to engineers doing CFD (Computational Fluid

Dynamics). These engineers typically have to use specialized software solutions to create geometry and discrete CFD models. Therefore they must learn two, three, four or more software solutions, user interfaces, tools and specialized rules.

Generally speaking, all engineers need to process geometry they receive as source data along with a simulation case description or a design specification. Sometimes they also need to create a spatial representation of the object based on drawings. This chapter shows how to handle geometry created by a structural engineer or a draftsman.

NX Advanced Simulation includes several commands for modifying (simplifying) the source geometry. These include commands on the **Model Preparation** toolbar such as *Idealize geometry*, *Defeature Geometry*, *Split Body*, *Sew*, *Divide Face*, as well as commands on the **Midsurface** toolbar: *Midsurface by Face Pairs*, *Offset Surface*, *User Defined Midsurface*, *Trim and Extend*, *Sew*.

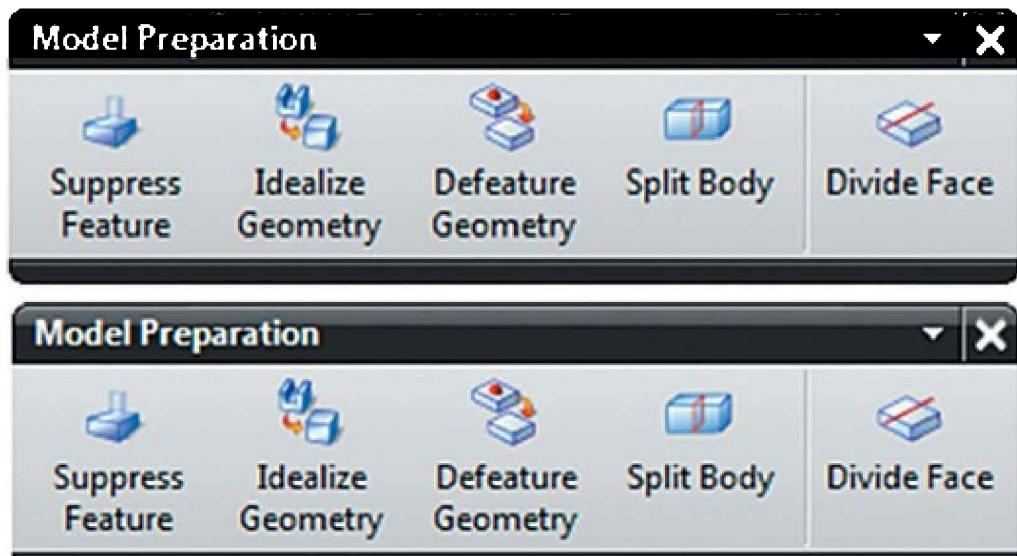


Figure 2.1. Model Preparation and Midsurface toolbars

You can also use **NX Advanced Simulation** to perform all synchronous modelling operations [1], which make adapting source geometry to a particular configuration substantially easier. You can find these commands on the **Synchronous Modeling** toolbar.



Figure 2.2. Synchronous Modeling toolbar

The computing engineer performs all geometry manipulation at the level of an ..._prt idealized geometry file. You can enable these commands in the idealized geometry file. To do this, use the **Simulation File View** window of the **Simulation Navigator** tab or choose **Window** in the main menu.

Note also that all past operations are stored in the model tree, and can be edited or deleted. To edit a prior operation, click the **Part Navigator** tab. Now double-click individual tree nodes or use their shortcut menus to return to a completed idealization operation at any time. You can also delete idealization operations by right-clicking them and choosing *Delete*.

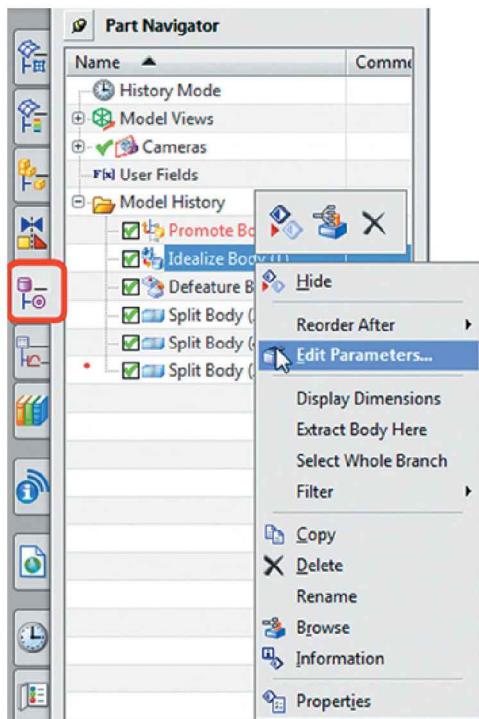


Figure 2.3. Part Navigator tab

2.2. Model Preparation toolbar commands

Consider each of the **Model Preparation** toolbar commands for geometry idealization.

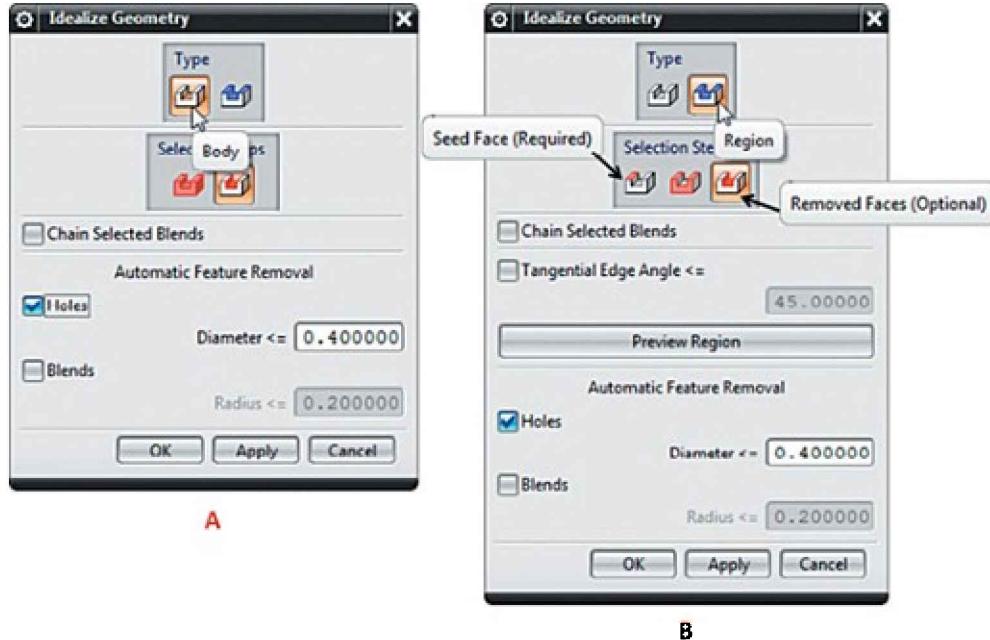
The **Idealize Geometry** command can automatically remove characteristic geometry objects of the model such as holes and blends. You can specify the diameter (or radius) of holes or blends as the characteristic parameter. You can use this command to select one body (solid body or sheet body) at the first step, and idealize it by specifying parameters described here (Figure 2.4, A).

At the second step you select the check box and specify a characteristic dimension (hole diameter or blend radius), or manually select faces for removal: *Removed Faces (Optional)*.

If the *Chain Selected Blends* option is selected, manually selecting a blend will extend the selection to all matched blends with the same radius until their radius changes or a bifurcation is encountered.

With the **Idealize Geometry** command you can also limit your work to a selected region. To do so, in the **Idealize geometry** dialog box select *Region* as the editing area type. You can select a *Seed Face* and *Boundary Face* that define the region where the idealization command operates (Figure 2.4, B).

Therefore with the **Idealize Geometry** command you can remove geometry elements that are not used in the simulation by manually selecting them or specifying a characteristic parameter.



*Figure 2.4. Idealize Geometry dialog box:
A. Body selection. B. Region selection*

Defeature Geometry strips geometry elements from the body. The main difference from **Idealize Geometry** is the ability to use Smart Selection. Smart selection provides the following options for this command: *Single Face, (Tangent Faces, Region Faces, Tangent Region Faces.*

After selecting faces for removal, click **OK**.

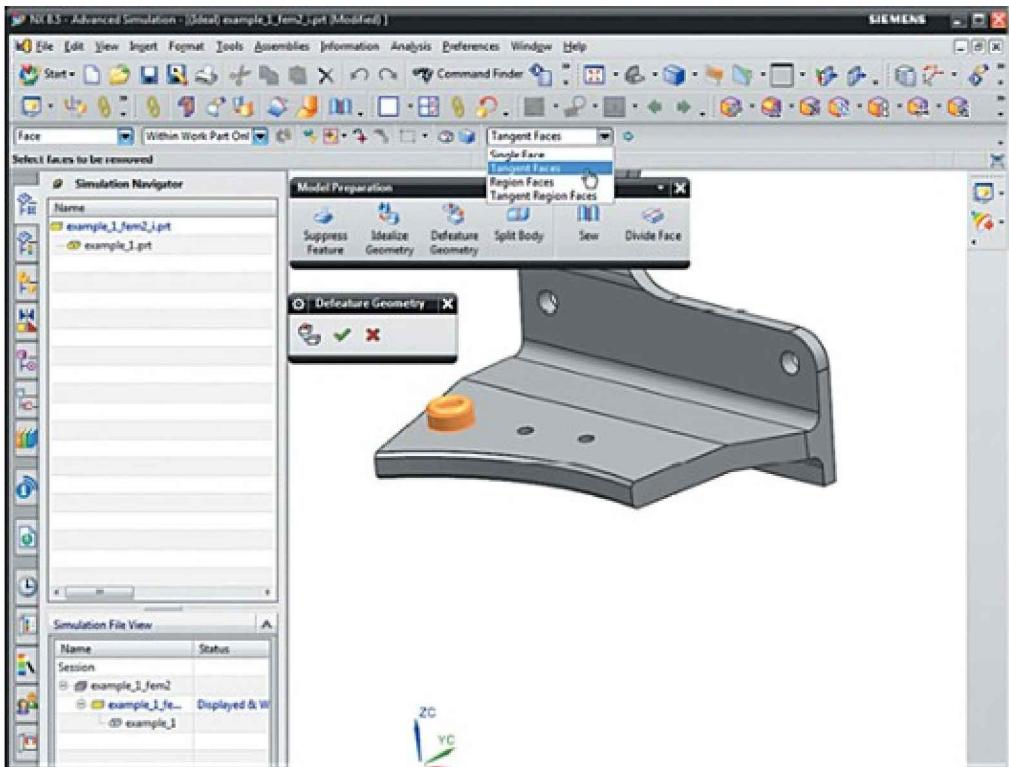


Figure 2.5. Defeature Geometry command

Split Body splits solid or sheet bodies into separate bodies. Both existing faces (face sets) and new or existing *Datum Plane* target planes can be used as the splitting tool. There are two other splitting tools that greatly enhance the capabilities of this command, namely extrusion and rotation surfaces (Figure 2.6).

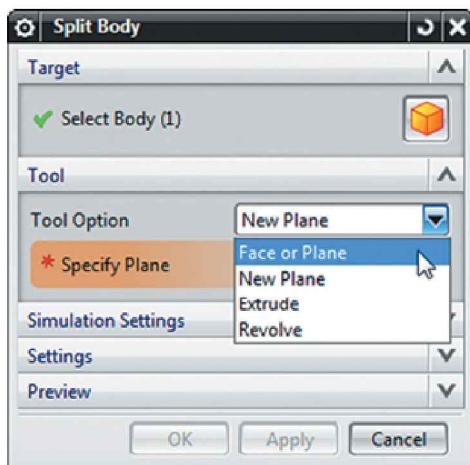


Figure 2.6. Split Body dialog box

Select the object (body) to be split, then specify the splitting tool.

If you select *Face or Plane* as the splitting tool, click an existing face (it must fully intersect the body being split) or an existing target sheet. This option is less frequently used in practice.

If you select *New Plane* as the splitting tool, you have to create a new drawing plate. You can choose one of the many available tools for creating new planes, which you can start by choosing the *Plane Dialog* command in the *Split Body* dialog box (Figure 2.7).

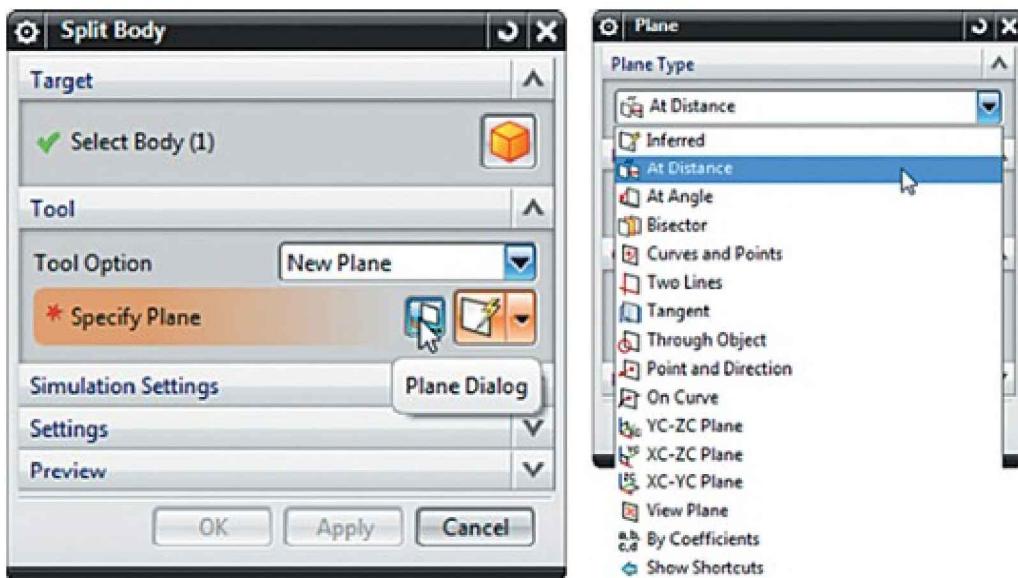


Figure 2.7. *Split Body* dialog box, *New Plane* selected as *Tool Option*

If under *Tool Option* you select *Extrude*, specify a curve or curve set that forms a section of the future splitting surface. Edges of the body itself or newly created curves can be used to specify the section. You can use curve drawing tools by choosing *Sketch Section* or (*Curve* in the *Section* area of the *Split Body* dialog box). Select the face on which to create the sketch section. After confirming the face selection, you are redirected to the 2D modelling environment (drawing tools are enabled), and the *Part Navigator* tab.

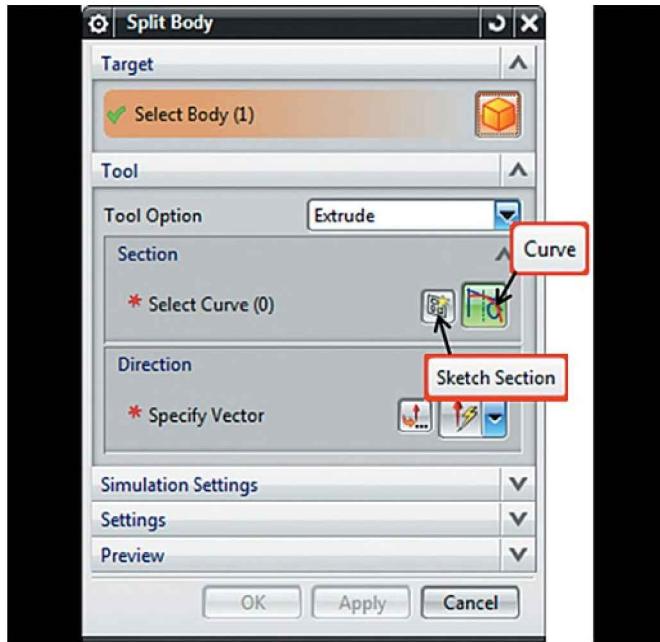


Figure 2.8. Split Body dialog box, Extrude selected as Tool Option

You also specify the extrusion direction (vector) of the selected section. There are two ways to specify the extrusion vector:

- Select a direction corresponding to the direction of an axis of the global coordinate system directly in the graphical area.
- Select a tool in the **Vector** dialog box by choosing the **Vector Dialog** menu command. or by selecting it in the tool drop-down list (Figure 2.9).

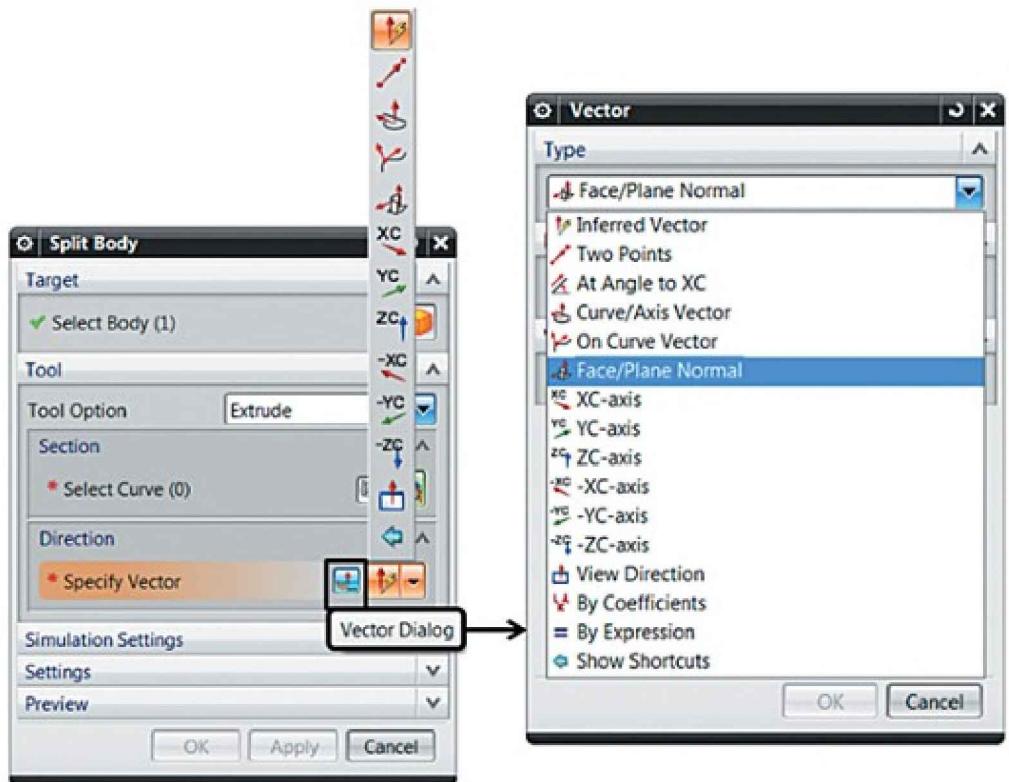


Figure 2.9. Split Body dialog box, specifying an extrusion vector

Note the created surface is infinite in the direction of extrusion. Keep this in mind when you plan the splitting operation.

When under *Tool Option* you select *Revolve*, you must specify a section curve for the future surface of revolution (it will be used as a body splitting tool). Edges of the body itself or newly created curves can be used to specify the section. You can create new curves using the same tools you use when under *Tool Option* you select *Extrude*. To complete the creation of the surface of revolution, specify the revolve axis by selecting or creating a vector and specifying a point on the revolve axis. The tools you use to create a vector that defines the orientation of the revolve axis, are similar to the tools used to define a vector in *Tool Option – Extrude*. You can specify a point on the revolve axis in two ways:

- Directly select a point in the graphics area after setting up the necessary selection references (end point, midpoint, arc centre, cross and others.)
- Use tools in the *Point* dialog box by choosing *Point Dialog* (Figure

2.10).

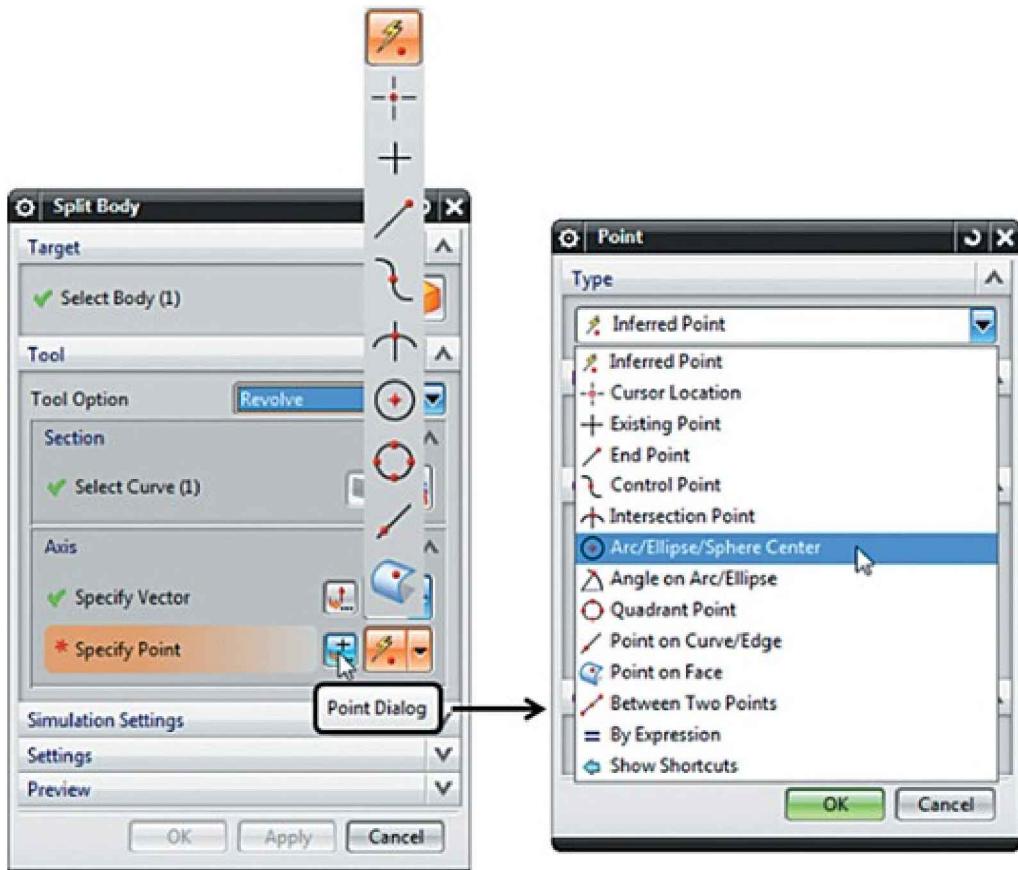


Figure 2.10. Split Body dialog box, tools for specifying a point on the revolve axis

The next group of the **Split Body** dialog box called *Simulation Settings* allows to set up special splitting options used only by simulating engineers. You can specify whether the body splitting process creates a mesh mating condition and node merging on adjacent faces (*Create Mesh Mating Conditions*).

Most importantly, the **Split Body** command allows preparing the body for creating the finite-element mesh based on hexahedral elements. You can use the *Check for Sweepable Body* check box to visually assess the possibility of creating a hexahedral mesh based on existing bodies. Red color indicates that building a hexahedral mesh is impossible, green color indicates that the body is prepared for building such a mesh, and amber color indicates that you need to further manipulate the geometry to attempt

building a hexahedral mesh. This option is useful as an automated assistant for deciding which type of finite elements to select, but the final decision should be based on engineering considerations and requirements for the quality of results, as well as available computing resources.

The *Keep Imprinted Edges* option allows to create a projection of an adjacent face (splitting face) of a split body onto another split body. From the engineer's perspective this option is closely related to the *Create Mesh Mating Conditions* option because it is used in practice to build a shared mesh on bodies produced by splitting with the **Split Body** command.

Show Result allows to preview the result of splitting before it is committed.



Figure 2.11. Split Body dialog box

You can use the **Divide Face** command to divide a face of a solid or sheet body using a line, a brim, a drawing plane, or another face.

In practice you can use this command to select a special area on a body face to control the quality of the finite-element mesh or load zone and constraints. If you use this method, you gain an advantage of not having to split the body, while still being able to divide its face. You can choose either the selected object itself or its projection on the face you want to divide as the *Dividing Object*. To do so, specify a *Projection Direction*. You would typically specify the direction along the normal vector to the surface (*Normal to Face*) or an existing (or newly created) vector (*Along Vector*). You can additionally hide the division tool when the operation is complete by

selecting *Hide Dividing Objects*.



Figure 2.12. *Divide Face* dialog box

Example 2.1. Using Model preparation toolbar commands

The usage of some commands on the **Model Preparation** toolbar can be demonstrated using the model shown in Figure 2.13 as an example.

1. Launch NX 8.5 and open *example_1.prt*, then go to **NX Advanced Simulation**: **Start**→**Advanced Simulation**.

Set default options for all dialog boxes. To do this, choose **Preferences**→**User Interface**→**General**→**Reset Dialog Box Settings** in the main menu then click **OK**.

Right-click the model on the **Simulation Navigator** tab (Figure 2.13). Choose *New FEM...* to create a FE model file. In the *FEM* file creation dialog box **New Part File**, specify the file name and directory, then click **OK**. In the *FEM* file creation dialog box **New FEM**, make sure *Create Idealized Part* is selected and click **OK**. Do not modify the rest of default settings in the **New FEM** dialog box.

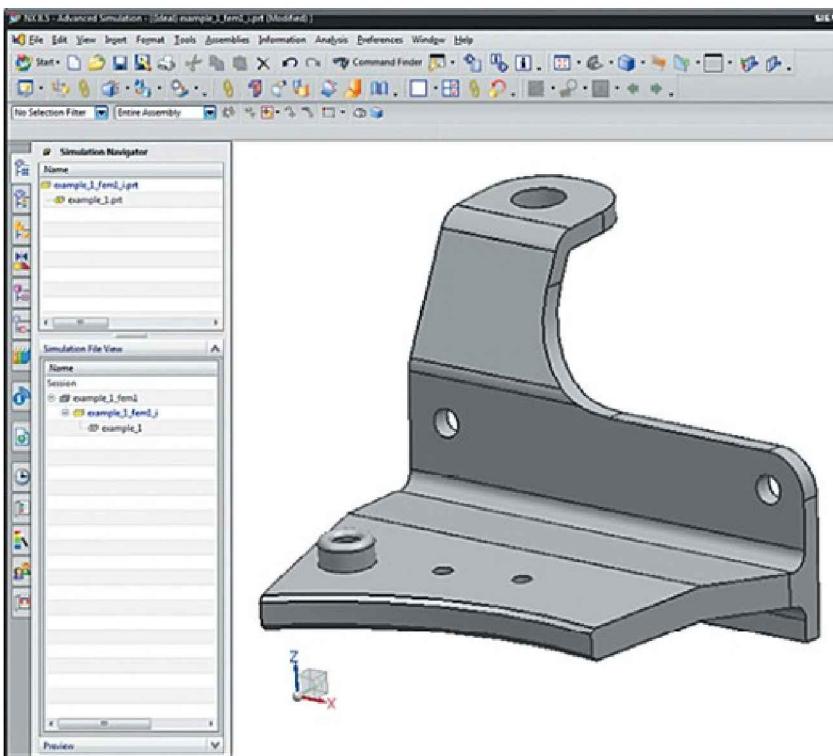


Figure 2.13. Creating idealized model and FE model files

You used the source CAD model to create files for the idealized model (*example_1_fem1_i.prt*) and the FE model (*example_1_fem1.fem*). Note that you can see the names of the three model files (Figure 2.13) in the **Simulation File View** window.

2. In this example only the idealized model is edited. Therefore, in **Simulation File View** window double-click *example_1_fem1_i.prt* (the active file name is highlighted in blue). A message appears suggesting you create an associative copy of the geometry model if you plan to perform any operations on geometry. Click **OK**.

Click **Promote** on the **Advanced Simulation** toolbar to create an associative copy.

3. Click **Idealize Geometry** on the **Model Preparation** toolbar. In the dialog box (Figure 2.14) do the following:
 - Select the part.
 - In the *Automatic Feature Removal* group, select *Holes and Blends*. In the *Diameter* box, type 0.2 mm, in the *Radius* box, type 0.3 mm. Note how all model objects that have characteristic values less

than the entered values are selected.

- Click **Apply**.
- In the **Selection Steps** area, click **Removed Faces** and select the remaining three holes by clicking them directly in the graphics area.
- Click **OK**.

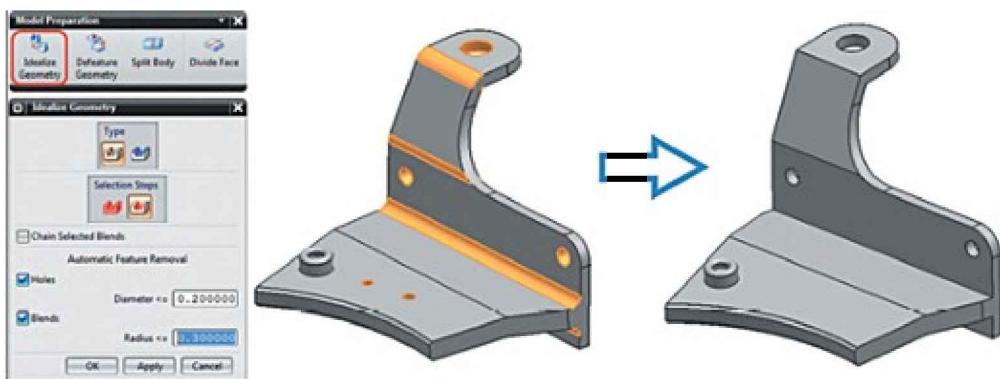


Figure 2.14. Idealize Geometry command, removing holes and blends

4. Click **Defeature Geometry** on the Model Preparation toolbar.

Set up *Face Rule* in the *Target Faces mode*. Then click a face of the flange to select all faces of this object.

In the **Defeature Geometry** dialog box (Figure 2.15), click the green check mark.

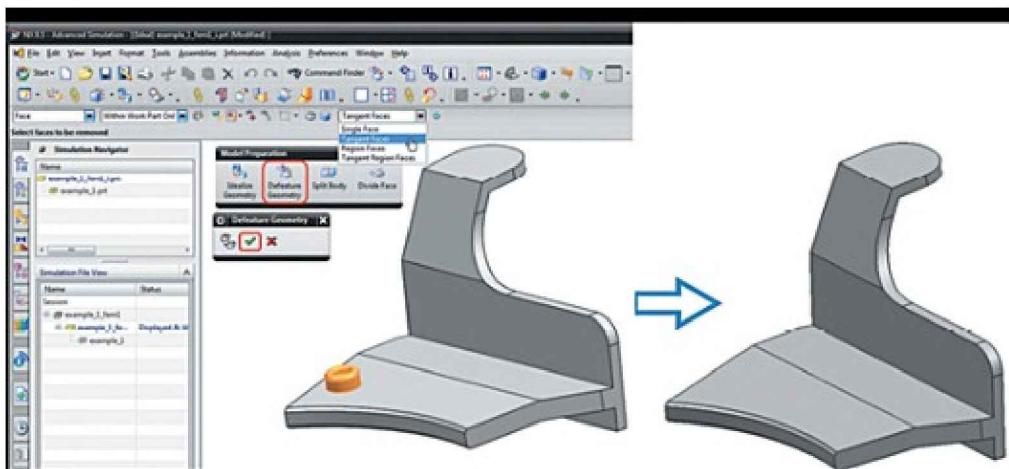


Figure 2.15. Defeature Geometry command, removing the “flange” model object

5. Choose **Split Body** on the Model Preparation toolbar.

In the **Split Body** dialog box (Figure 2.16), do the following:

- Select the part.
- Under *Tool Options* choose *New Plane*. Select *At Distance* from the list of plane creation methods to create a plane at a specified distance from an existing face.
- Specify a vertical face as shown in Figure 2.16. The selected face is highlighted, and a blue arrow appears, pointing along the normal vector of the selected face, as well as a distance indicator.
- Hold the left mouse button on the blue arrow and drag the new plane to extend it to the end point of the edge as shown in Figure 2.16.
- Click **Apply**.

The **Split Body** dialog box stays open.

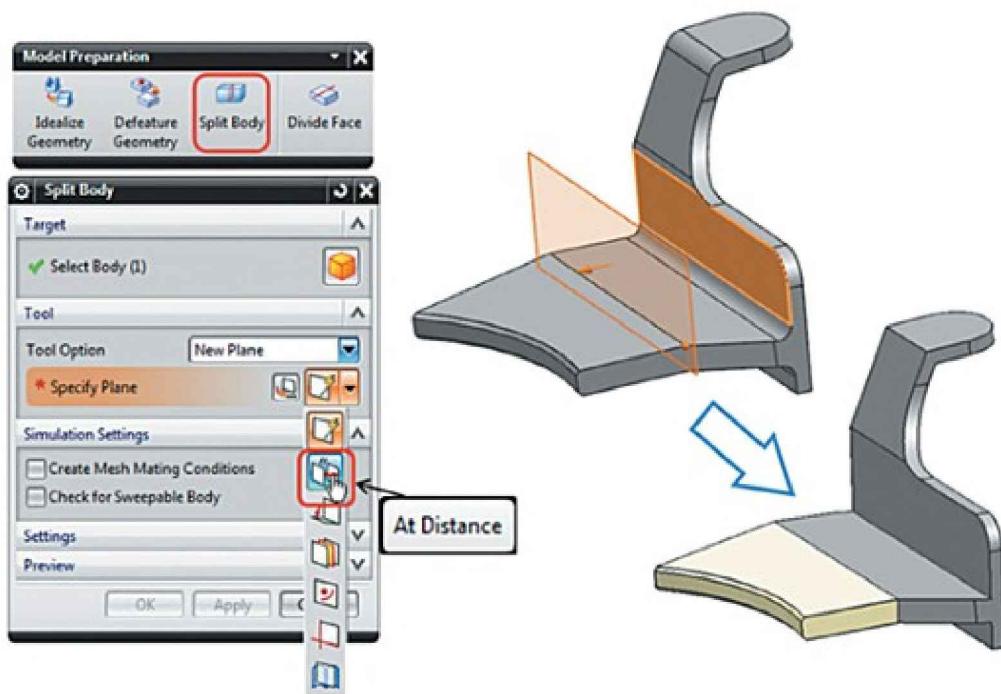


Figure 2.16. Split Body command, Tool Options, New Plane

In the **Split Body** dialog box, do the following:

- Select a body.

- Under *Tool Options* select *Extrude*.
- In the *Section* group select the body edge highlighted in green in Figure 2.17 as the sectional curve.
- In the *Direction* group select *Two Points* in the vector tools drop-down list. Select the points as shown in Figure 2.17.
- Click **Apply**.

The **Split Body** dialog box stays open.

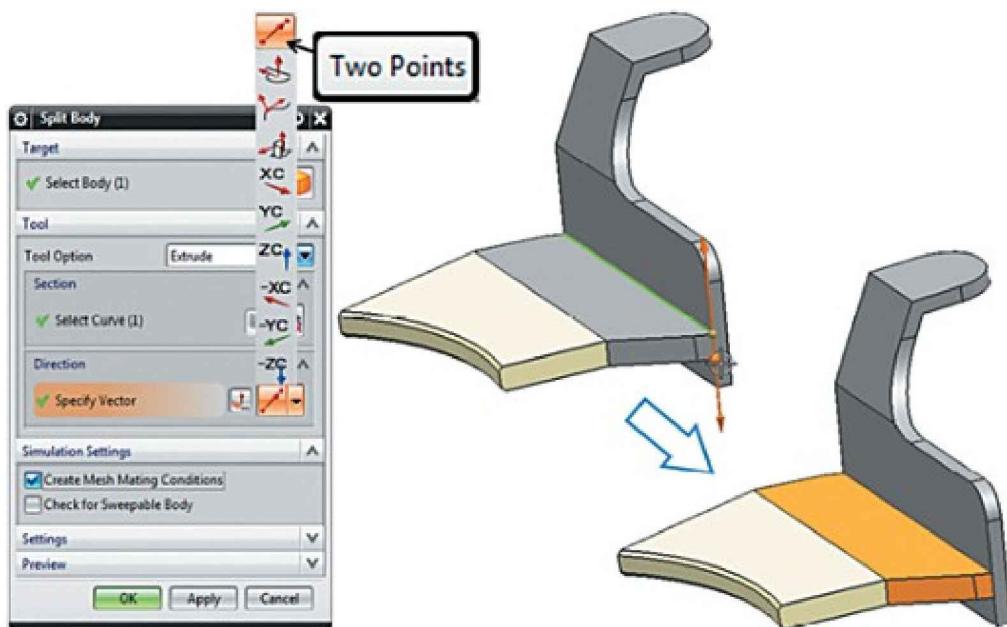


Figure 2.17. Split Body, Tool Options, Extrude

In the **Split Body** dialog box, do the following:

- Select a body.
- Under *Tool Options* select *Extrude*.
- In the *Section* group, for the *Curve* option select a face of the body to create a sketch as shown in Figure 2.18, A. Note that you will automatically switch from **NX Advanced Simulation** to **NX Sketch**, which offers a broad selection of drawing tools. Use the *Line tool* to create a line by two points as shown in Figure 2.18, B. Click *Finish Sketch*. You will automatically return to **NX Advanced Simulation**.
- In the *Direction* group, select the direction suggested by the command (along the normal vector of the selected face).

- Click **OK**. The result is shown in Figure 2.19.

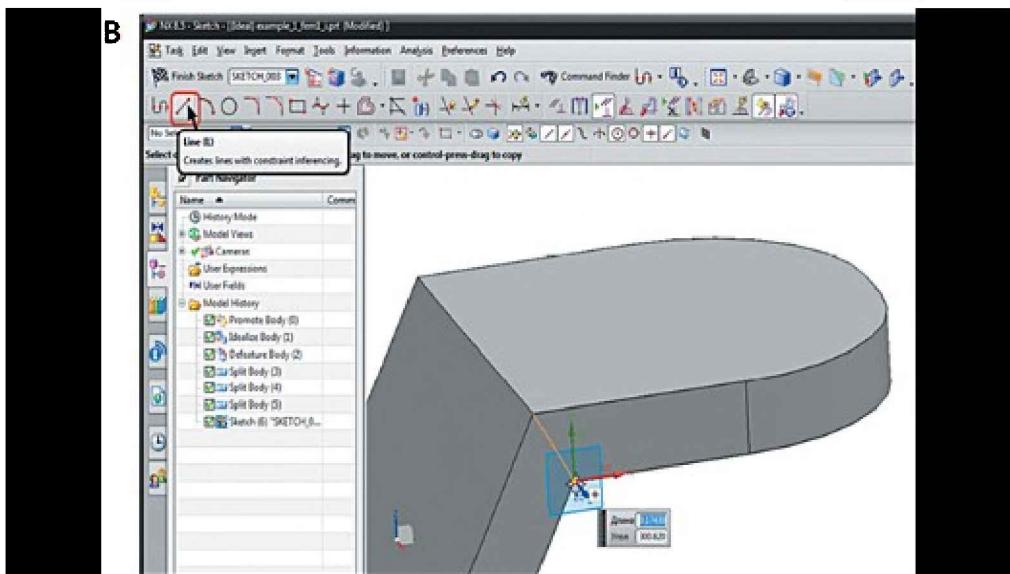
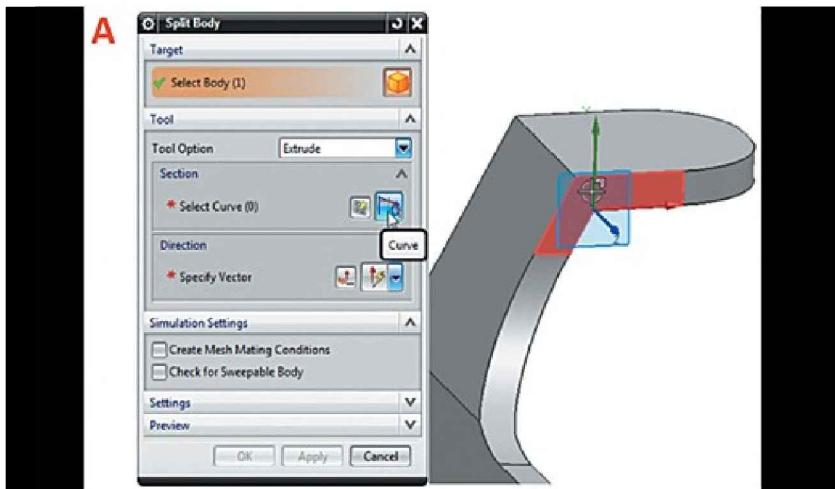


Figure 2.18. Split Body, Tool Options, Extrude:

A. Specifying the plane. B. Drawing a sketch

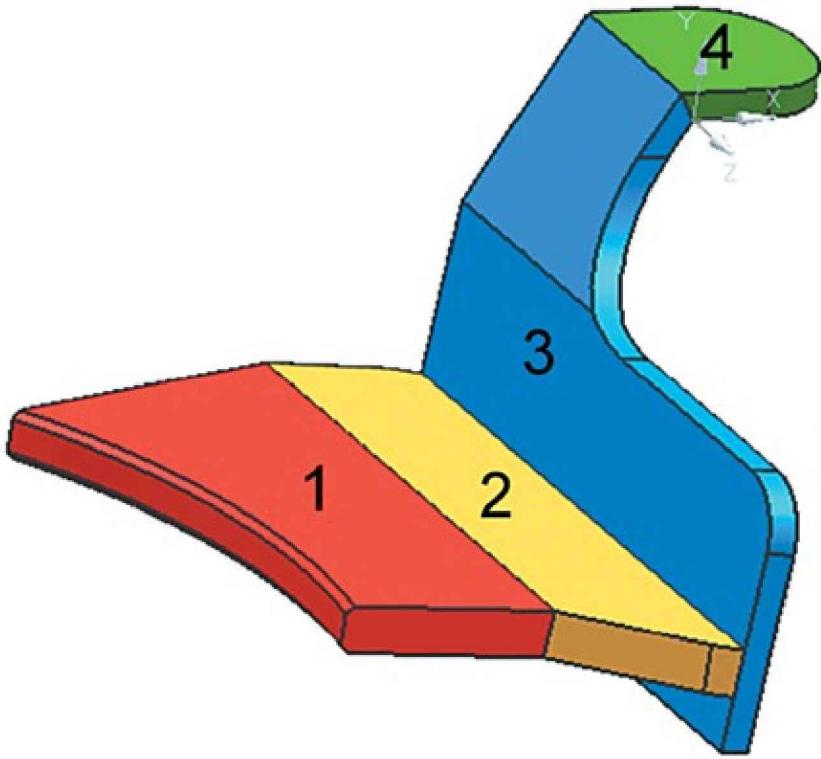


Figure 2.19. The result of model geometry simplification using Model Preparation toolbar commands

To save the model, choose **File→Save All** in the main menu.

2.3. Midsurface toolbar commands

One of the most common techniques for thin shell structure creation is to use a midsurface. Note that solid body spatial models created for thin shell structures show great precision only when finely meshed. To guarantee practical convergence of results, you must strictly follow the rules of creating such finite-element meshes. Often to create a shell model is a better solution. To do this, choose the **Midsurface by Face Pairs** command. To use this command, select the body (or several bodies) to build a midsurface. After selecting the bodies in the *Faces to Exclude from Pairing* group of the **Midsurface by Face Pairs** dialog box, you can specify the faces that will be excluded from the automatic search of face pairs for midsurface creation (Figure 2.20).



Figure 2.20. *Midsurface by Face Pairs* dialog box

As soon as the bodies are selected, in the ***Midsurface by Face Pairs*** dialog box the *Face Pairing*, group is activated, where you can select a body face search strategy (method) for creating midsurfaces. NX Advanced Simulation provides three midsurface creation strategies: *Progressive*, *Thickness*, and *Manual*. With the first method you can create midsurfaces virtually automatically: the system selects the average thickness of the model as the thickness of the thin shell structure to be “replaced” with a midsurface. With the second strategy you can find all pairs of surfaces separated by a distance smaller than the specified value. Midsurfaces are built when you click *Automatically Create Face Pairs*. The third strategy allows you to fine-tune midsurface creation in completely manual mode. With the *Manual* strategy you can select one face and use the *Side 2 Search*, option specifying a *Search Distance* value to find a complementary face. To use an existing sheet body as the midsurface, click *Select Sheet Body or Face* on the *Replacement Mid-Sheet* tab. Select two faces manually and click *Create Face Pair* to create a pair for the midsurface.

Regardless of the strategy of choice, all pairs you create are displayed in the *Face Pair List* group. If you need to modify a face pair, you only need to

select the pair from the list, and the strategy will automatically change to *Manual*, irrespective of the initially selected strategy. Additional options in *Advanced Trimming Tools* and *Advanced Midsurface Tools* groups will become active as well. Options in the *Advanced Trimming Tools* group include:

- *Trim to Side 1 Faces* trims the midsurface along the contour of the first face.
- *Trim to Solid Body* trims the midsurface along the boundary of the solid body.
- *Standard Trimming* trims the surface based on both (the first and the second) faces of the pair. Options in the *Advanced Midsurface Tools* include:
 - *Use Side 1* creates a midsurface on the first face of the pair.
 - *Offset* specifies a midsurface by offsetting the first face of the pair to the midsurface.
 - *Cloud of Points* creates a midsurface based on a point cloud created between the faces of the pair.
 - *Standard* creates a midsurface using the most suitable method for the given pair.

When building midsurfaces for complex parts (group of parts, assembly) to use display options for original bodies, face pairs and resulting midsurfaces can be helpful. In the **Midsurface by Face Pairs** dialog box, you can set the following *Display Options*:

- *Hide Solid Body Upon Apply* hides the original bodies after building midsurfaces.
- *Show Pairing Faces As Transparent* makes the faces in a pair transparent throughout the construction process and when using the command.
- *Show Midsheets As Transparent* makes the midsurfaces transparent.
- *Refresh Display* updates the displayed content.

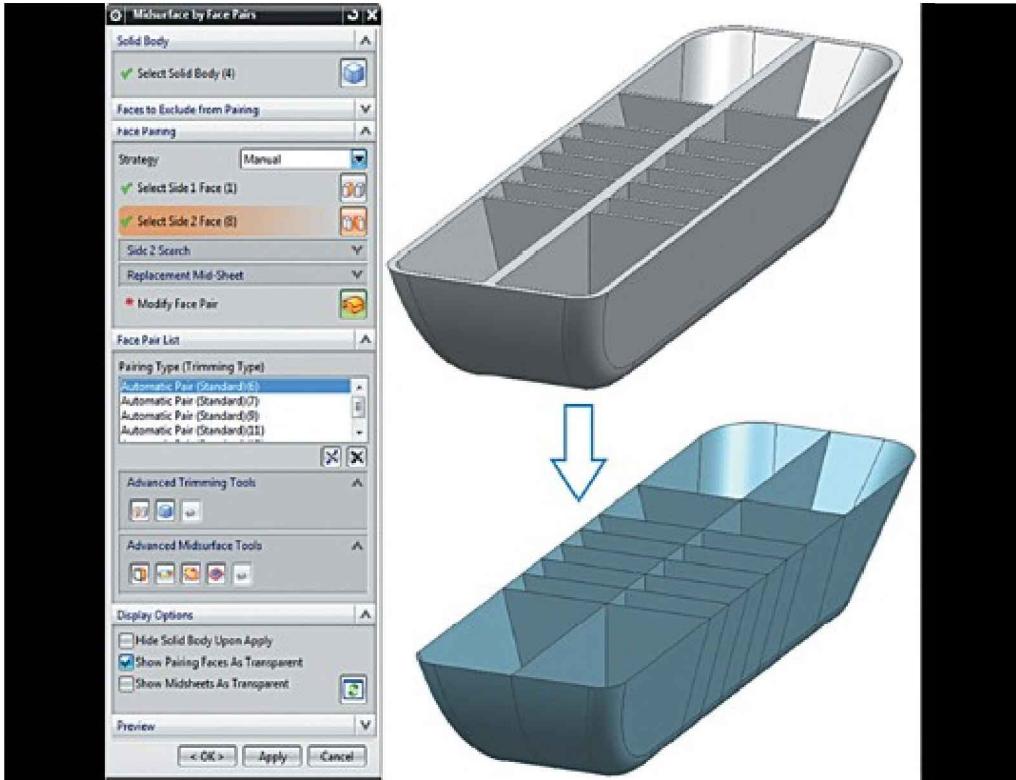


Figure 2.21. *Midsurface by Face Pairs* dialog box

Example 2.2. Using the Midsurface by Face Pairs command

The usage of ***Midsurface by Face Pairs*** can be demonstrated using the model discussed in the previous example.

1. Opening a CAD model and creating new model files
Repeat steps 1 and 2 of Example 2.1.

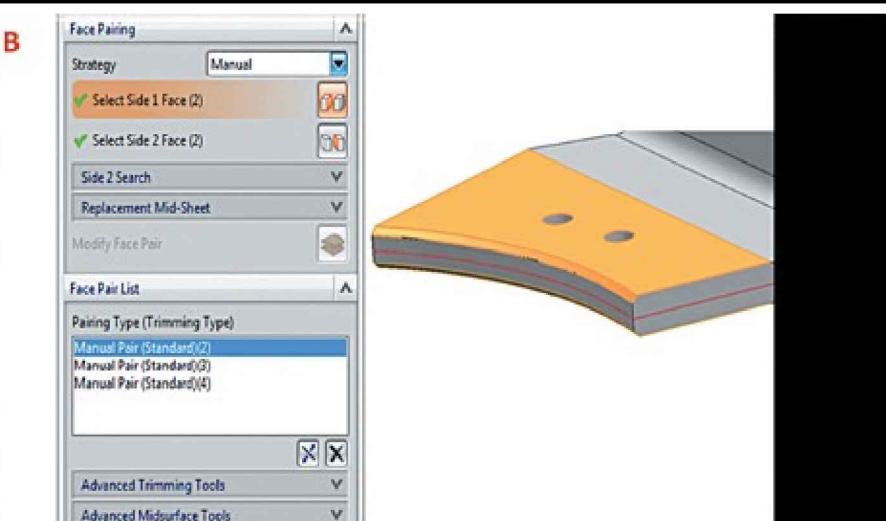
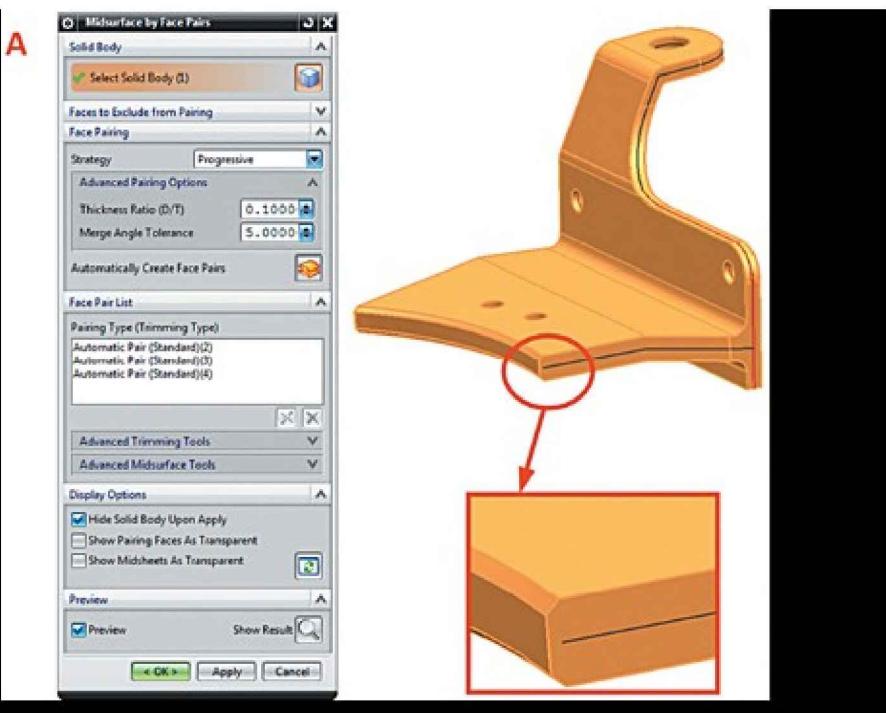
2. ***Midsurface by Face Pairs***

Click ***Midsurface by Face Pairs*** on the **Midsurface** toolbar. In the ***Midsurface by Face Pairs*** dialog box (Figure 2.22), do the following:

- Select the part.
- In the *Strategy* list select *Progressive*.
- Click *Automatically Create Face Pair*. Note that all face pairs you create are listed in the *Face Pair List* group.
- Under *Display Options* select *Hide Solid Body Upon Apply*. Note that the midsurface in the chamfer region (Figure 2.22, A) is not

complete. To remedy this, add the faces that form the chamfer to *Automatic Pair (Standard)* face pair (2). Click this pair under *Face Pair List* to automatically change the strategy to *Manual*. Then add a corresponding face to each side (faces of side 1, faces of side 2) (Figure 2.22, B).

- Click *Modify Face Pair*. The name of this face pair changes from *Automatic Pair (Standard)(2)* to *Manual Pair (Standard)(2)*.
- Click **OK**. The result is shown in Figure 2.22, B.



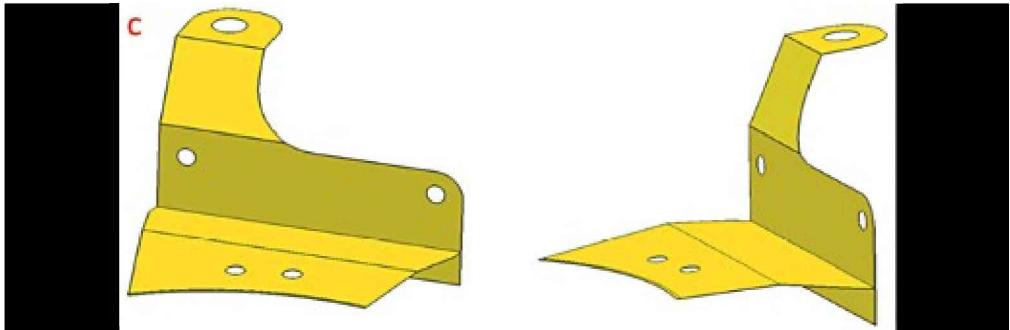


Figure 2.22. Creating midsurfaces:
A. Creating face pairs automatically. B. Fixing a face pair manually. C. Result

You can use the **User Defined Midsurface** command to specify any sheet body as the midsurface of a solid body. You can choose both the solid body and the surface. This command is practical when standard and advanced methods of midsurface creation cannot be used or do not achieve a satisfactory result.



Figure 2.23. User Defined Midsurface dialog box

You can control the thickness of the midsurface in this way: if the nodes of the finite-element mesh lie inside the body, then the thickness (calculated based on the thickness of the body) can be closed within the boundaries from *Inside Minimum* through *Inside Maximum*. If the nodes lie outside the solid body, their thickness is equal to the thickness specified in the *Outside* box.

It is important to note that for all methods of midsurface creation, the thickness of shell-type finite elements is calculated automatically. To do this, after creating a 2D mesh on the midsurface, click **Edit Mesh Associate Data** and in the **Mesh Associated Data** dialog box, change the value of *Thickness Source* from *Physical Table Property* to *Midsurface*.

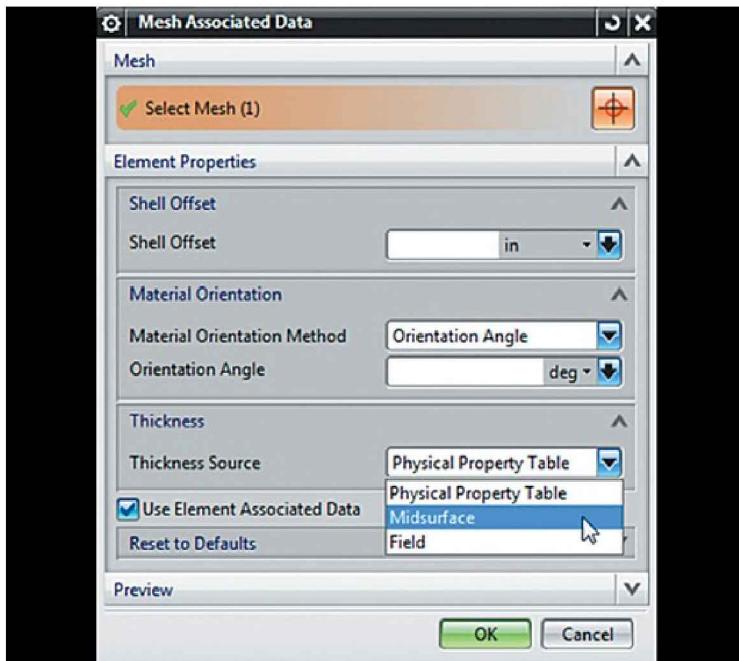


Figure 2.24. Mesh Associated Data dialog box

With the **Offset Surface** you can create an offset copy of a surface. If the offset is measured from the midsurface, the offset surface also inherits the thickness information.

With the **Sew** command you can join sheet bodies as well as solid bodies. This command is most often used to sew the created midsurfaces into a single surface. Note that only sheet bodies are sewn after joining, and the number of faces stays the same. To sew sheet bodies, in the *Type* list select *Sheet*, and select one *Target* sheet body and one or more *Tool* sheet bodies. To work with solid bodies (in the *Type* list select *Solid*), select adjacent faces of neighbour bodies. These faces are then joined.

You can use the **Trimmed Sheet** command to trim a part of the sheet body using curves, faces, edges or the coordinate plane as the tool. Start by selecting the sheet body that you need to trim, then select the tool directly.

In some cases the new midsurfaces intersect in places where it is not acceptable. The **Trimmed Sheet** command is very useful in solving problems of this kind.

Use the **Trim and Extend** command to trim or extend a face (or face set) by a given distance or until it intersects with another face (or face set). You will often encounter large gaps between midsurfaces oriented at an angle relative to each other. Use the **Trim and Extend** command to easily solve such problems when creating midsurfaces.

Example 2.3. Using Midsurface toolbar commands

The usage of **Midsurface** toolbar commands can be demonstrated on the model created in Example 2.1 (Figure 2.19).

1. Launch NX 8.5 and open *example_1_fem1_i.prt* idealized geometry file, switch to NX Advanced Simulation: **Start→Advanced Simulation**.

Set default options for all dialog boxes. To do this, choose **Preferences→User Interface→General→Reset Dialog Box Settings**, click **OK**.

2. Click **Midsurface by Face Pairs** on the **Midsurface** toolbar. In the **Midsurface by Face Pairs** dialog box (Figure 2.25) do the following:

- Select body 1.
- Under *Strategy* select *Manual*. Select faces for side 1 and side 2 (Fig. 2.25, A).
- Click **Create Face Pair**.
- Under *Display Options* activate *Hide Solid Body Upon Apply*.
- Click **Apply**.
- Select body 2.
- Under *Strategy* choose *Thickness*, and enter 7 mm (Figure 2.25, B).
- Click *Automatically Create Face Pair*.
- Under *Display Options* activate *Hide Solid Body Upon Apply*.
- Click **OK**. The result of the operations is shown in Figure 2.23, C.

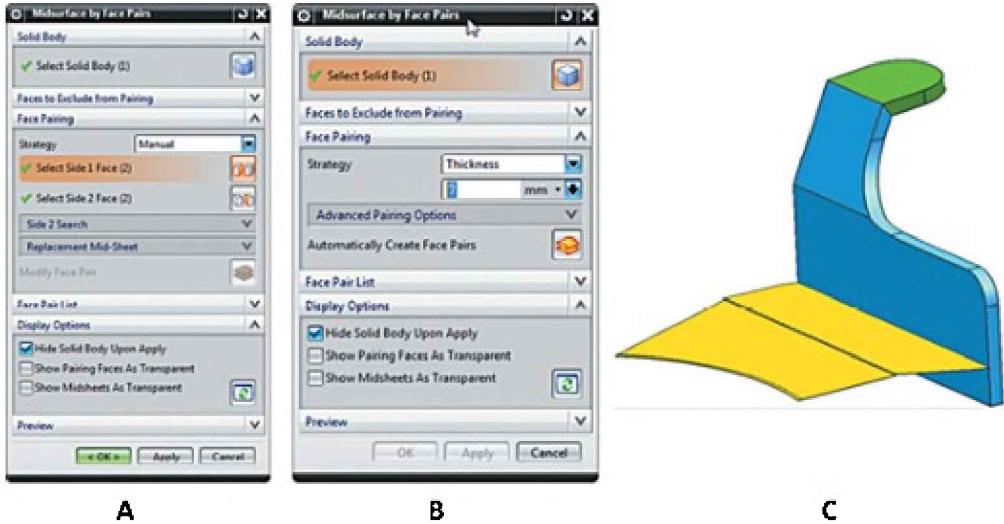


Figure 2.25. Creating midsurfaces:

A. Manual strategy. B. Thickness strategy. C. Result

3. Click **Offset Surface** on the **Midsurface** toolbar. In the **Offset Surface** dialog box (Figure 2.26), do the following:

- Select faces of bodies 3 and 4, as shown in Figure 2.26.
- Enter **Offset1** value of 2 mm.
- Click **OK**.

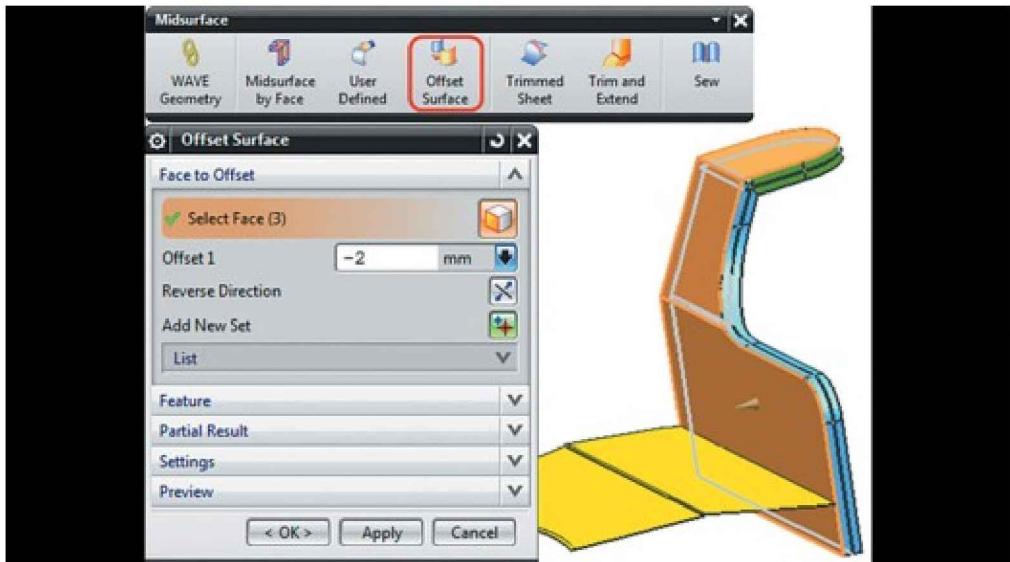


Figure 2.26. Creating midsurfaces using the Offset Surface command

The result of the operations is shown in Figure 2.27. Note that a number of

problem areas exist in the model: intersection of midsurfaces, misalignment, and gaps. You will often encounter such situations when representing a source model using midsurfaces.

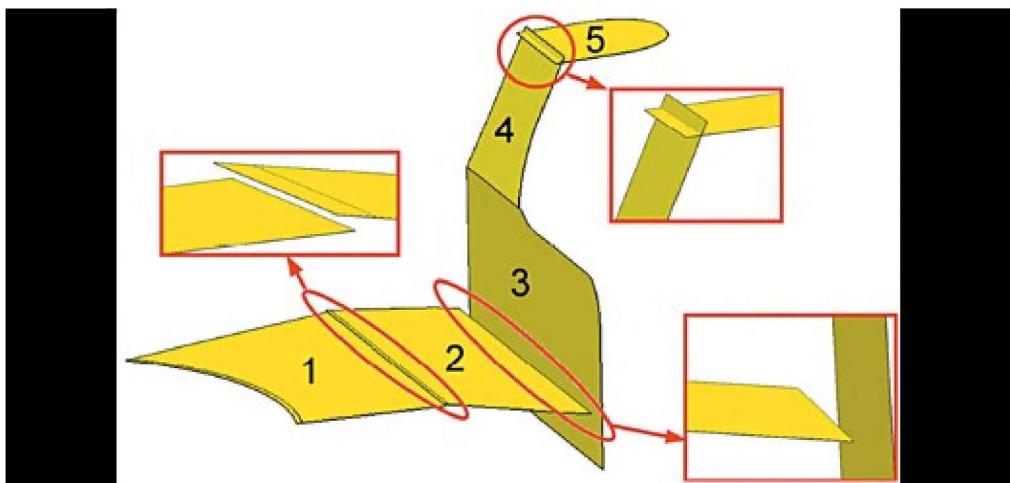


Figure 2.27. Geometry requiring elimination of problem areas

4. Click **Trim and Extend** on the Midsurface toolbar. In the **Trim and Extend** dialog box (Figure 2.28), do the following:
 - Under *Target* select the right-hand edge of face 2.
 - Under *Tool* select face 3.
 - Click **OK**.

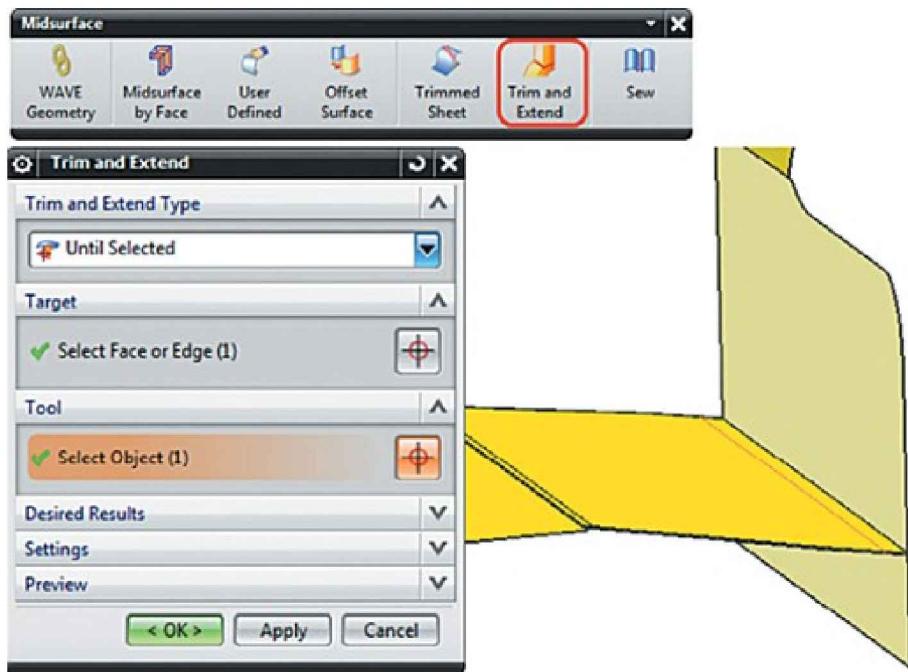


Figure 2.28. Trim and Extend

5. Click **Sew** on the **Midsurface** toolbar. In the **Sew** dialog box (Figure 2.29), do the following:
 - Under *Type* select *Sheet*.
 - Under *Target* select face 1.
 - Under *Tool* select face 2.
 - Under *Settings* enter a *Tolerance* value of 0.6 mm.
 - Click **OK**.



Figure 2.29. Sew command

6. Click **Trimmed Sheet** on the **Midsurface** toolbar. In the **Trimmed Sheet** dialog box (Figure 2.30), do the following:

- Under *Target* select face 4.
- Under *Boundary Objects* select face 5.
- Under *Projection Direction* select *Normal to Face*.
- Under *Region* select *Keep*.
- Click **Apply**. The **Trimmed Sheet** dialog box stays open.

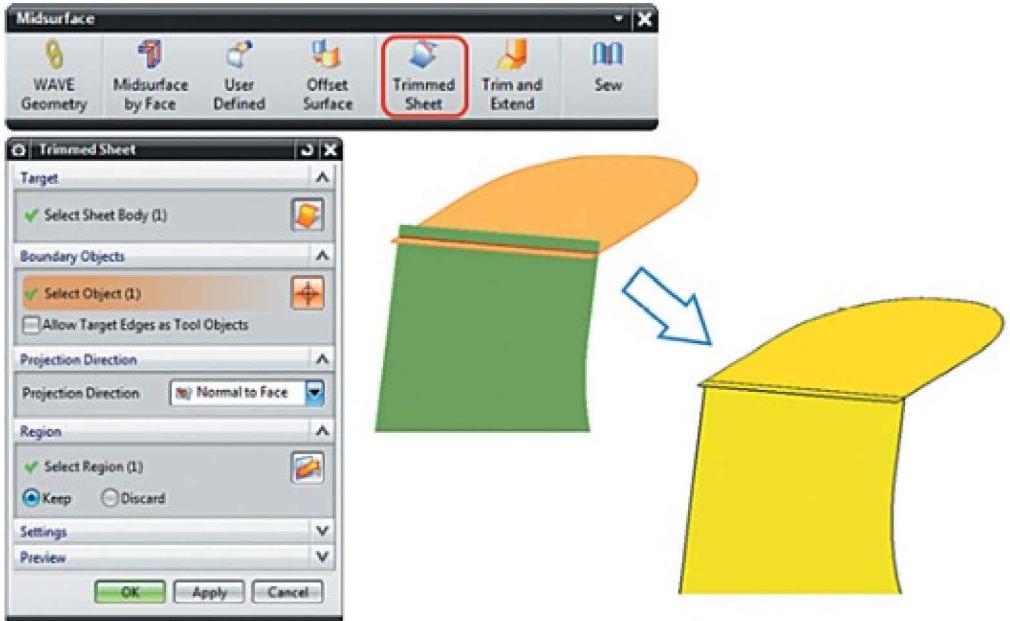


Figure 2.30. Trimmed Sheet command

- Under *Target* select face 5.
- Under *Boundary Objects*) select face 4.
- Under *Projection Direction* select *Normal to Face*.
- Under *Region* select *Keep*.
- Click **OK**.

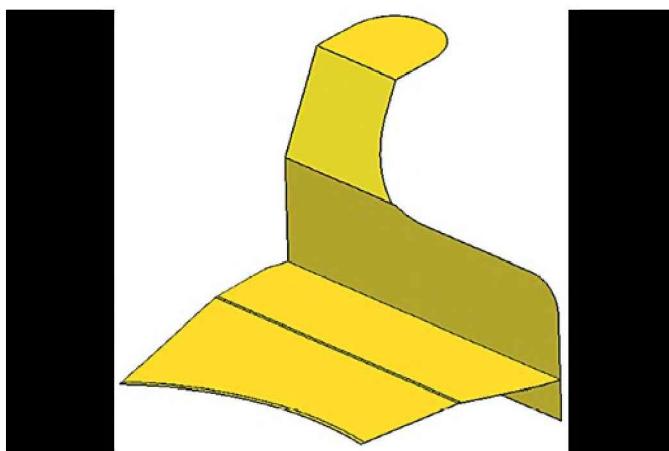


Figure 2.31. Results of using the Midsurface toolbar commands

2.4. Commands of the Synchronous Modeling toolbar

In NX 6 and later you can use commands of the synchronous modeling

toolbar. The terms “Synchronous Technology” and “synchronous modeling” mean extending the parametric modeling approach with direct modeling technology to enable more efficient and intuitive user interaction with the geometry model even in case the geometry is provided history-free with no parameters. These terms also imply the ability of the mathematics kernel to recognize existing geometry connections and determine dependencies in real time. Synchronous modeling provides a set of tools that allow to modify the model by directly selecting and modifying faces, edges, and vertices while remaining agnostic about the exact software that was used to create the CAD model. To learn more about synchronous modeling as implemented in NX **design system**, refer to [1].

NX Advanced Simulation is a full-featured CAE environment within the united NX design platform. This means an engineer can use all tools of the CAD system of NX, including synchronous modelling. Using synchronous process tools, engineer can independently introduce modifications into the geometry model, including modifying outline dimensions and local dimensions, deleting geometry elements, moving and rotating faces, refining and modifying parameters and locations of holes, shifting edges and sets of any other elements together (regardless of intersections in the resulting geometry). In fact, the computing engineer introduces the necessary modifications into the geometry data to create a structure that satisfies simulation criteria regardless of the way the original geometry data were created or the choice of the initial design system. The tools for automated updating of the finite-element mesh in accordance with modifications of geometry can still be used (these are discussed in the following chapters).

The synchronous process combines parametric modelling based on structural bodies with tools for direct editing of geometry. The synchronous process is made possible by a variational approach to product design, which involves binding boundary elements of a geometry model with logical and parametric constraints, both user-defined and automatically recognized by the system.



Figure 2.32. Synchronous Modeling toolbar

The synchronous process was developed on the basis of Parasolid geometry kernel and a set of D-Cubed integrated modules. Using the synchronous process workflow, you can create and edit models element-by-element without a build history (that is, without using elements in a build tree). The solver synchronizes existing elements (geometry) and interrelations between elements with a build history, the character of modifications, and parameters (links). With this approach you can take geometry in a neutral format or geometry from other CAD systems and process it in the same way as you would process geometry created in NX. The synchronous modelling commands themselves are stored in the simulation model build tree, where you can edit or delete them.

The most important synchronous modelling commands used in practice in NX Advanced Simulation are: **Move Face**, **Replace Face**, **Resize Blend**, **Resize Face**, **Pull Face**, **Delete Face**, **Cut Face**, **Copy Face**, **Make Coplanar**, **Make Coaxial**, **Make Tangent**, **Make Parallel**, **Make Perpendicular**, **Make Symmetric**, **Linear Dimension**, **Angular Dimension**, **Radial Dimension**. This book discusses the most useful (frequently used) commands of the **Synchronous Modeling** toolbar specifically from the computing engineer's perspective.

You can use the **Move Face** command to modify part geometry by moving selected faces to a distance or angle using dimension or geometry links with adjacent faces. Moreover, you can move faces by directly entering dimensions into the part. The command allows intersection of objects after moving, moving whole objects or parts of objects, and moving objects outside the part outline (that is, actually removing the object from the part). You can also use the **Move Face** command as a simple geometry modification tool while doing design work in other NX applications that use

the created master models, such as tooling design or CNC programming. To use this command, on the **Synchronous Modeling** toolbar, click **Move Face** or choose main menu command **Insert → Synchronous Modeling → Move Face**. In the **Move Face** dialog box, select faces for moving in the **Face** group. In this dialog box under **Face Finder** on the **Results** tab, you can find parameters for automatic search of link between faces (such as *Coaxial*, *Equal Radius*, *Offset*, *Coplanar Axes*, and others.).

The *Motion* option in the *Transform* group is set to *Distance-Angle* by default, therefore the Steering Wheel shows both axis markers and plane markers. You can also specify the following parameters for the *Motion* option: *Distance* for incremental offset of the face, *Angle* for rotation of the face, *Distance Between Points* for measuring the face offset distance relative to existing geometry objects, *Radial Distance* for measuring offset distance along a radius, *Point to Point* for moving by a vector drawn from one point to the other, *Rotate by Three Points* to rotate the face by three points, *Align Axis to Vector* to rotate the face around an axis and measure the rotation angle by two vectors.

You also need to specify the movement direction using the *Specify Distance Vector* option. There are two ways to specify the vector:

- Select a direction corresponding to the direction of an axis of the global coordinate system directly in the graphical area.
- Select a tool in the **Vector** dialog box by choosing the **Vector Dialog** menu command or in the drop-down list. You can use the **Settings** group to set up the *Overflow Behavior* option, which specifies face movement characteristics in case it intersects other faces when adding material volume (movement that creates new geometry as opposed to deleting existing geometry). You can select one of four methods for the *Overflow Behavior* option:
 - If you select *Automatic*, the parameter can influence both the surface being moved and the intersected surface depending on the situation.
 - If you select *Extend Change Face*, the face being moved is changed depending on the geometry of encountered faces.
 - If you select *Extend Incident Face*, the face being moved is extended to fit with the encountered face.
 - If you select *Extend Cap Face*, the movement is carried out by

simple addition of material (with the edges of the face being moved overhanging the others).

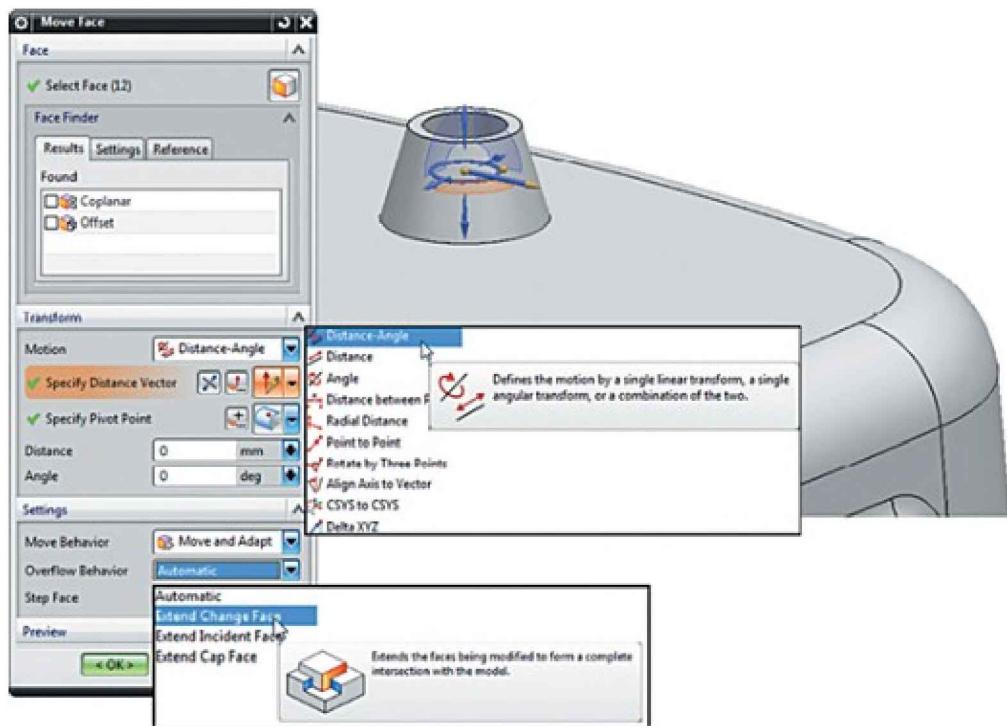


Figure 2.33. Move Face command

You can use the **Resize Face** command to change a radius (of one or more objects), one or several angles. Using *Smart Selection* together with this command allows to quickly adjust the model in accordance with the chosen criteria. In the *Size* group the system automatically determines the current diameter of the cylindrical face in the *Diameter* box. You can also enter a new value to modify the diameter.



Figure 2.34. Resize Face command

You can use the **Resize Blend** command to change the radius of one or more blends. You can also combine this command with the Smart Selection. You can enter the new blend radius in *Radius* box.



Figure 2.35. Resize Blend command

You can use the **Delete Face** command to delete a part of an object or the whole object. This command restores breaks in the model that result from using the command. You can combine this command with *Smart Selection* to quickly delete or edit several objects or an entire object in a single operation. In the *Type* group, in addition to the familiar *Face* option you can also select *Hole* to delete holes from the model by specifying their radial dimension or by selecting the holes directly in the graphical area.

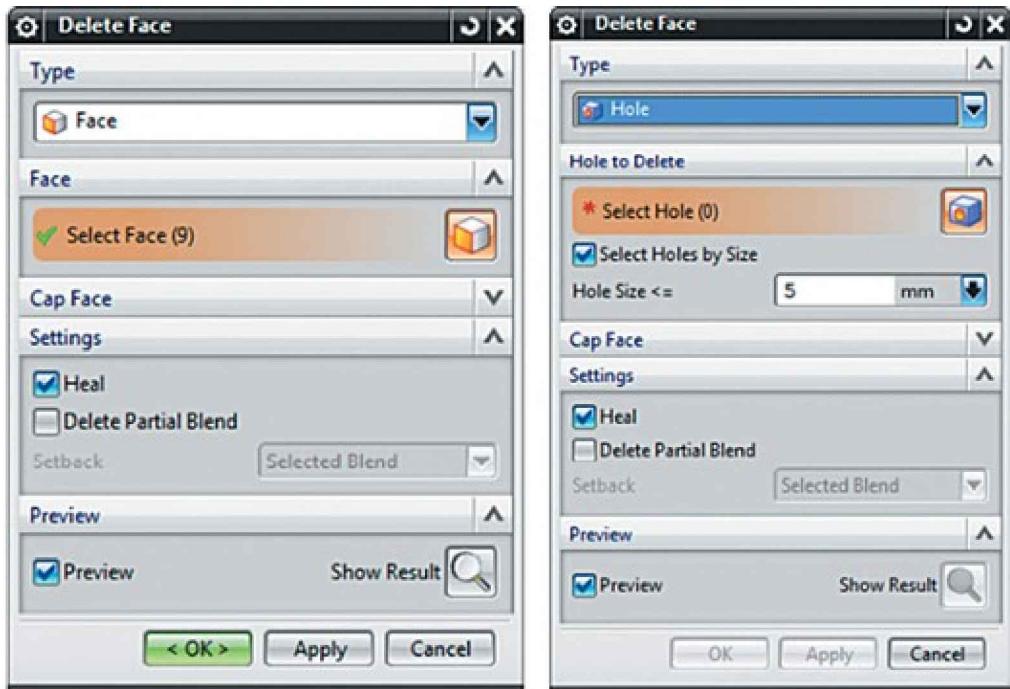


Figure 2.36. Delete Face command

In practice the **Delete Face** command is used when the **Idealize Geometry** command does not achieve the desired result and vice versa, that is, these commands work well in combination.

The **Replace Face** command replaces one or more existing faces with one or more replacement faces. A replacement face can come from the same part or from a different body. After replacing the selected face with a different face, the system automatically matches the face with adjacent faces and blends. The **Face to Replace** group includes all faces that you need to replace. The **Replacement Face** group specifies faces that are used as replacement. You can use the **Offset** option in the **Replacement Face** to specify additional linear offset applied to the face after it is replaced. You can use the **Overflow Behavior** option in the **Settings** group to specify options for face rebuilding when the face intersects another, similar to other synchronous modelling commands.



Figure 2.37. Replace Face command

You can use the **Pull Face** command to build new parts. Use it to quickly build faces and bodies based on existing geometry without creating additional sketches. You can use this command to build one or more geometry elements at the same time. In the *Face* group select the face to extend in accordance with the *Motion* option value: by a certain *Distance*, by a *Distance between Points*, by a distance and direction between two points (*Point to Point*).

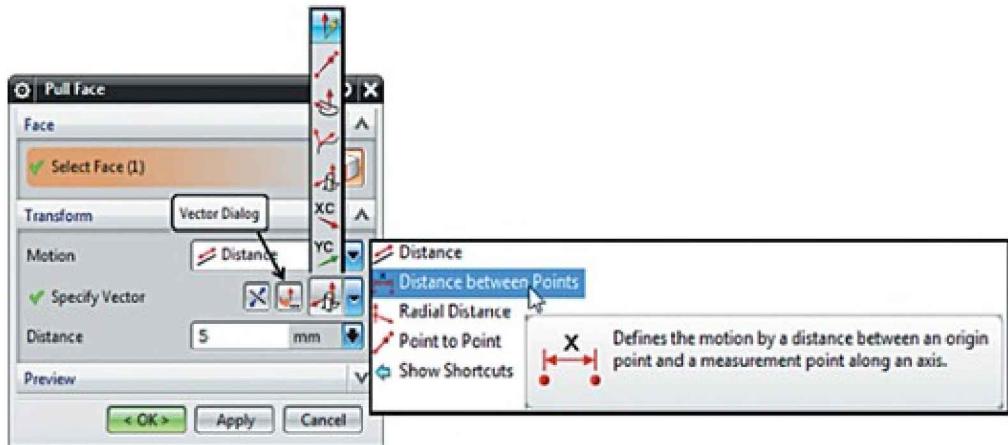


Figure 2.38. Pull Face command

Commands for copying and cutting faces: **Copy Face**, **Cut Face**, **Paste Face**, **Mirror Face**, **Pattern Face**.

Commands for applying geometry conditions: **Make Coplanar**, **Make Coaxial**, **Make Tangent**, **Make Parallel**, **Make Perpendicular**, **Make**

Symmetric. These commands operate on two faces: the mobile face that you need to transform and the immobile face that you need to match with the mobile one.

Commands for modifying the model by adding dimensions: **Linear Dimension**, **Angular Dimension**, **Radial Dimension**. When you add a dimension or change its value, you modify the face it was created for. A computing engineer can substantially increase efficiency by adding driving dimensions directly to the model in 3D mode regardless of the original geometry format. The most important options of these commands are the following:

- *Select Origin Object* to specify the object of origin for the linear dimension.
- *Select Measurement Object* to definitively specify the linear dimension.
- *Select Face* to select a face (or face set) to move when the linear dimension changes, and options to select several faces in the *Face Finder* group.

It is also important to note that the synchronous solver allows bidirectional editing of geometry. This means that elements can be changed irrespective of creation order. In this way links between elements such as “parent-child” are eliminated both within a single part and in the whole assembly.

In addition to the main stages of numerical analysis, the synchronous process can substantially improve efficiency of specialized problems such as building models for fluid dynamics analysis or optimization analysis. For example, solving an optimization problem involves building a parametric model (in essence, recreating the original geometry) or imposing additional sensitivity or optimization parameters on the structural engineer. These are the factors that substantially complicate such calculations. As a result, engineer makes a number of calculation iterations while varying a certain parameter. With synchronous modelling tools you can specify a defining outline dimension, a blend or hole radius, or a stiffener position to be used as optimization parameters. In this way, the simulating engineer can transform any simulation model into an optimization model in only a few operations. The format of source geometry data is not important in this case as well.

In fluid dynamics problems the **Delete Face** synchronous modelling tool allows to create a fluid domain geometry region in one operation. This domain can then be prepared for building a structured simulation mesh or automatically split into tetrahedral elements with a prismatic layer (for an in-depth discussion, see Chapter 6, part 2).

Another important factor to consider when working with mathematical model creation tools (idealization and synchronous process tools) is their associativity as well as the ability of NX Advanced FEM preprocessor (NX Advanced Simulation without including the NX Nastran solver) to automatically recognize all modifications and update the FE mesh. The simulation mesh is rebuilt, while the loads, constraints, and other parameters are maintained. This feature of NX Advanced Simulation combines with idealization and synchronous modelling tools to allow modifying the source geometry while tracking changes and modifying the simulation model correspondingly.

Example 2.4. Using Synchronous Modeling toolbar commands

Usage of some commands on the **Synchronous Modeling** toolbar can be demonstrated using the model shown in Figure 2.39 as an example. You must first create a FE mesh to demonstrate its automatic updating by synchronous modelling tools after modification of geometry.

1. Opening a CAD model and creating new model files

Repeat steps 1 and 2 of Example 2.1.

You used the source CAD model to create an idealized model file (*example_2_fem1_i.prt*) and an FE model file (*example_2_fem1.fem*). Note that you can see the names of the three model files (Figure 2.39) in the **Simulation File View** dialog box. In the **Simulation File View** dialog box, double-click *example_2_fem1.fem* to activate it (the active file name is shown in blue).

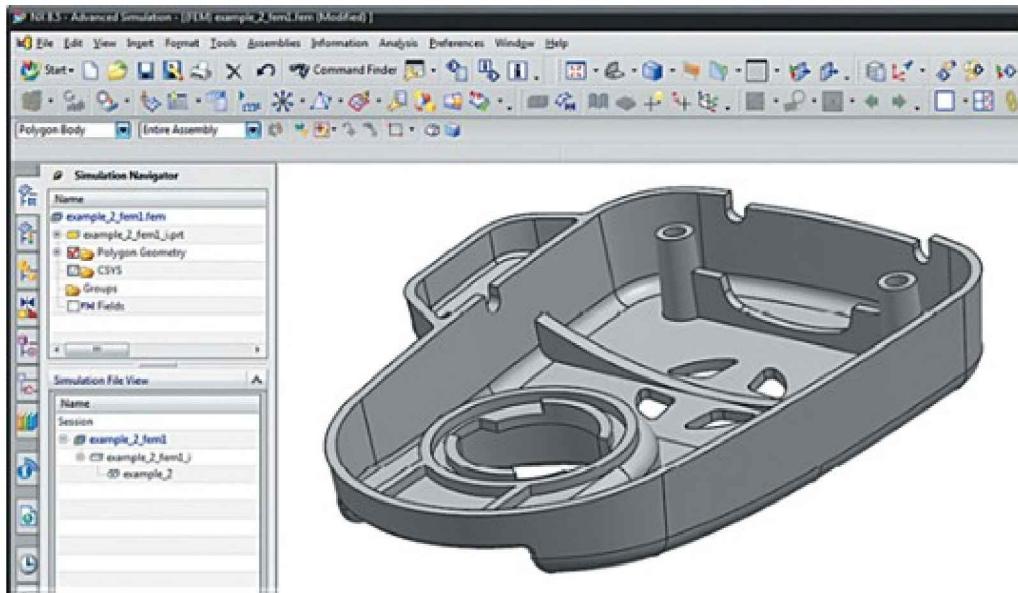


Figure 2.39. Creating idealized model and FE model files

- To create the mesh, click **3D Tetrahedral Mesh** on the Finite Element Model toolbar (Figure 2.40):

- Click **Select bodies** to select the part.
- Under **Type** select **CTETRA(4)**.
- Under **Element Size** select **Automatic Element Size**.
- Keep default values of the rest of the **3D Tetrahedral Mesh** dialog box.
- Click **OK**.

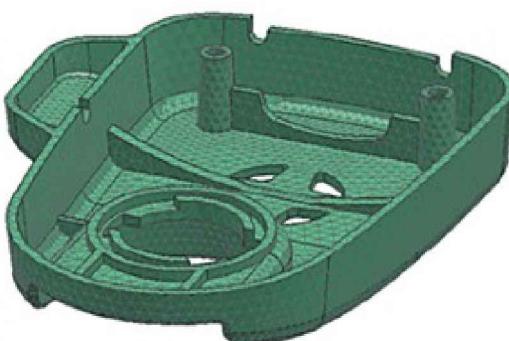


Figure 2.40. Creating the FE mesh

- In the **Simulation** **File** **View** **dialog** **box**, double-click

`example_2_fem1_i.prt` to activate it (the active file name is shown in blue). Click **Move Face** on the **Synchronous Modeling** toolbar. In the **Move Face** dialog box, (Figure 2.41) do the following:

- Select the faces highlighted in Figure 2.41, A.
- In the *Motion* list select *Distance*.
- Under *Specify Vector*, select a direction corresponding to the OY axis of the global coordinate system in the graphical area (the axis arrow changes to a conical shape) (Figure 2.41, B).
- In the *Distance* box enter 25 mm.
- Click **OK**.

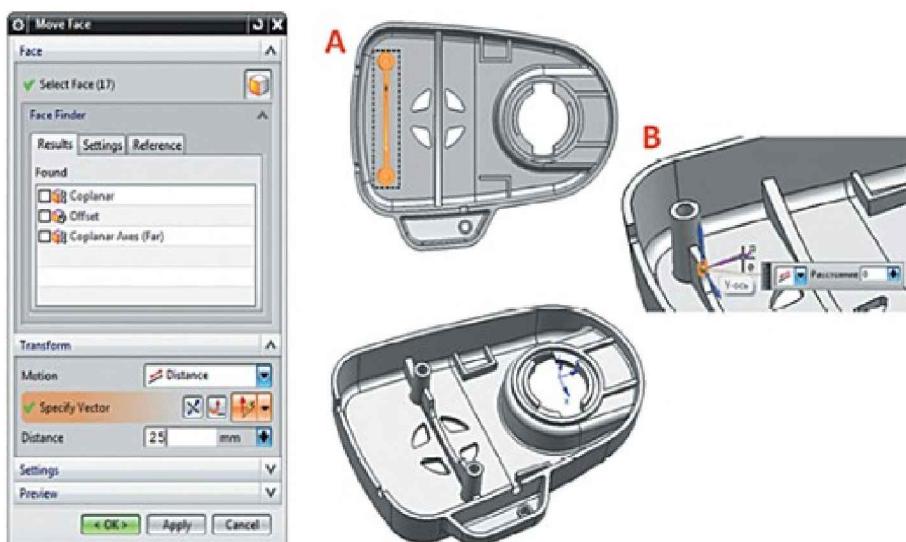


Figure 2.41. *Move Face* command

4. Click **Resize Face** on the **Synchronous Modeling** toolbar. In the **Resize Face** dialog box (Figure 2.42), do the following:

- Select the faces highlighted in Figure 2.42.
- In the *Diameter* box enter 22 mm.
- Click **OK**.

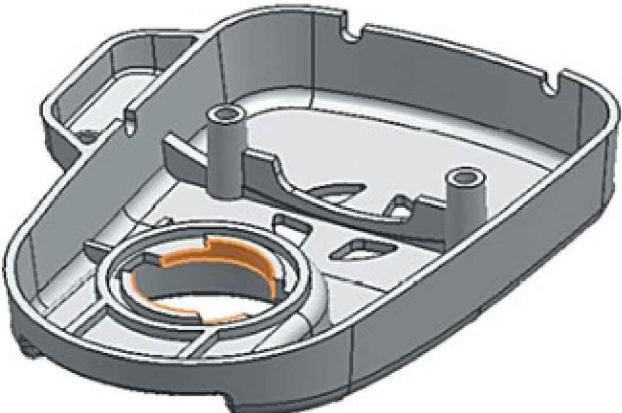
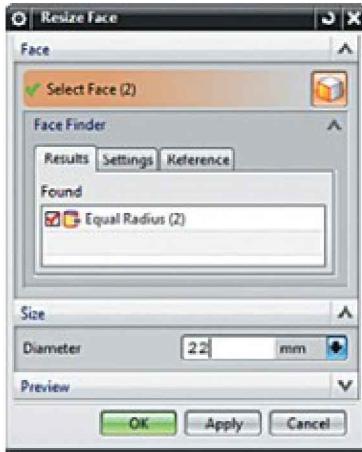


Figure 2.42. Resize Face command

5. Click **Offset Region** on the **Synchronous Modeling** toolbar. In the **Offset Region** dialog box (Figure 2.43), do the following:
 - Select the 3 faces highlighted in Figure 2.43.
 - In the *Distance* box enter 2 mm.
 - Click **OK**.

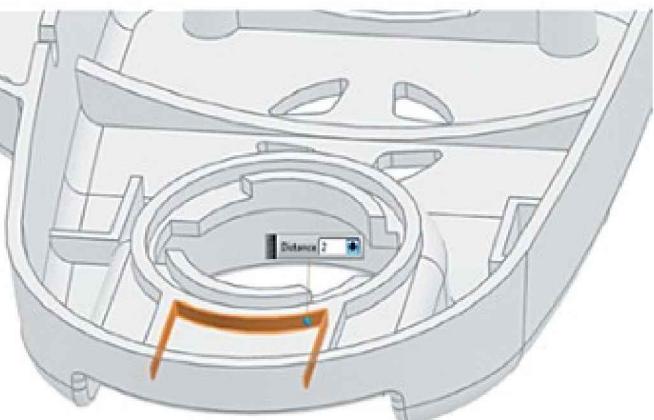


Figure 2.43. Offset Region command

6. Click **Make Coplanar** on the **Synchronous Modeling** toolbar. In the **Make Coplanar** dialog box (Figure 2.44), do the following:
 - Under *Motion Face* select face 1.
 - Under *Stationary Face* select face 2.
 - Under *Motion Group* select face 3.
 - Click **OK**.

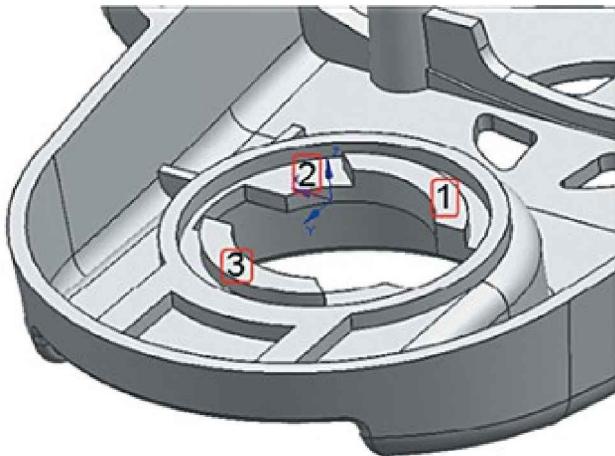


Figure 2.44. Offset Region command

7. Click **Delete Face** on the **Synchronous Modeling** toolbar. In the **Delete Face** dialog box, (Figure 2.45) do the following:
 - Select geometry elements highlighted in Figure 2.45.
 - Click **OK**.

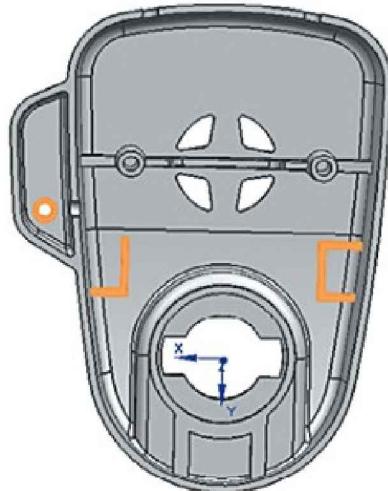


Figure 2.45. Delete Face command

This concludes the modification of source geometry. It should also be noted that synchronous process tools facilitate modification of geometry. All completed operations are reflected in the model tree on the **Part Navigator** tab (Figure 2.46). All modifications are introduced into the idealized model (*example_2_fem1_i.prt*), while the source master model (*example_2.prt*)

remains unchanged.

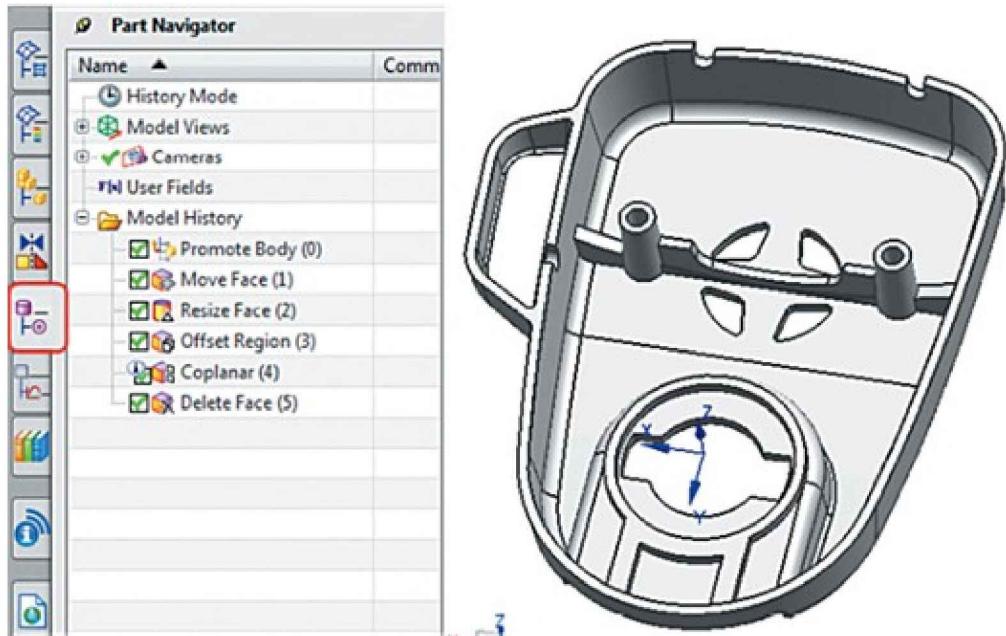


Figure 2.46. General view of the modified model

8. Update the FE mesh.

In the **Simulation File View** dialog box, double-click `example_2_fem1.fem` to activate it (the active file name is shown in blue). Note the regions with modified geometry. The model used to build the FE mesh is no longer up-to-date, so the system needs to update the FE mesh to reflect the modified geometry. Click **Update FE model** (Figure 2.47).

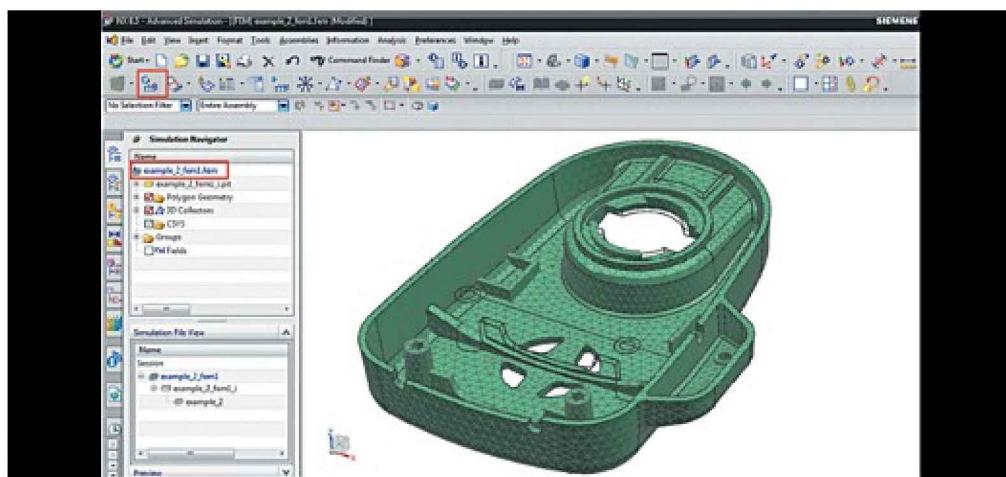


Figure 2.47. Update the FE mesh.

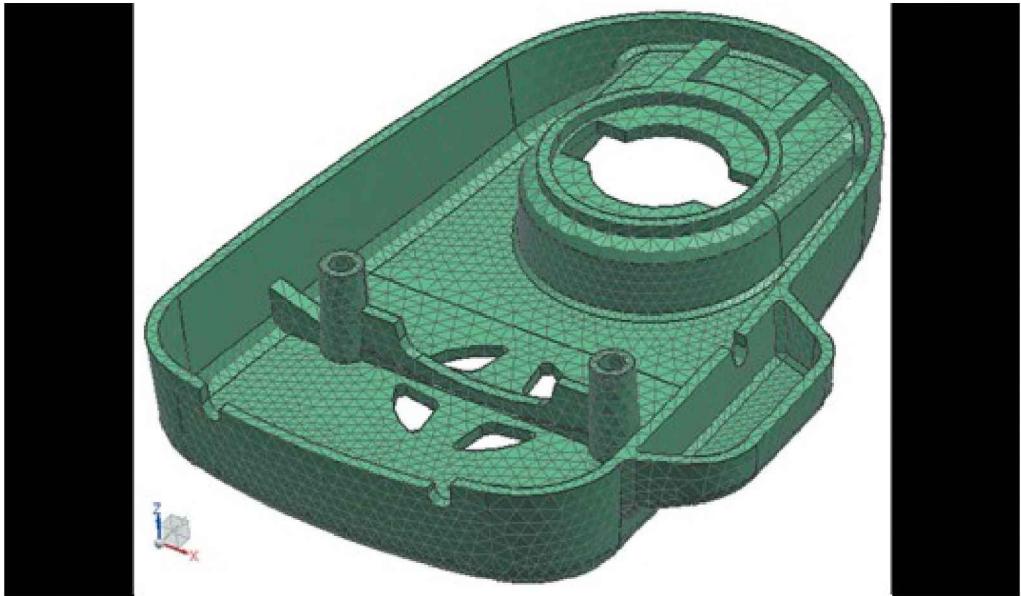


Figure 2.48. FE mesh update result

The FE mesh is rebuilt, reflecting all modifications of geometry.

Chapter 3. Creating and manipulating finite-element models

After creating an idealized geometry model, the next task is to design a high-quality and representative finite-element model. This stage of the engineering analysis workflow is critical and typically the most labor-intensive. The FE model (mesh) quality should never be neglected because if you have to work with a low-quality mesh (large elements, degenerate element shape) you can encounter divergence or a drop in the quality of the results. The degree of discretization, however, should be kept reasonable because otherwise the cost in computing resources and time can be considerable. When you work with a FEM file, in addition to building the FE model itself you also define physical properties of the model such as material properties and parameters of shell and rod elements. A *name_fem.fem* file is created for the default FE model, where “name” is the name of the source master model.

The newly created FEM file contains polygon geometry. Therefore, when you create an FE mesh, all mesh parameters and geometry settings are applied to the polygon geometry, not the idealized model. The polygon geometry is, however, associatively linked to the idealized geometry.

Figure 3.1 shows the **Advanced Simulation** toolbar with commands for working with materials, collectors, and meshes, commands for healing polygon geometry, and others.



Figure 3.1. Advanced Simulation toolbar commands, FE model

3.1. Structure of the FE model. Simulation Navigator

Before you proceed to study the methods and special aspects of creating FE models in NX Advanced Simulation you need to familiarize yourself with the **Simulation Navigator** tab of the Resource Bar. This tab is a tree view of the current model that contains links to all objects in the model. **Simulation Navigator** content changes when you go from the FE model (FEM Part) to the simulation model (Simulation Part) and vice versa.

Here is a more detailed description of the **Simulation Navigator** for a typical FE model. You can use it to manage the most important FE data:

- View FE model structure and content.
- Quickly access the idealized model or the master model.
- Create new objects such as new meshes, groups, and coordinate systems.
- Manage display options of various objects in the graphics window.
- Edit, rename, and delete existing objects.

Figure 3.2 shows a typical FE model tree with objects organized in nodes:

- *Polygon Geometry* (list of all polygon bodies of the model).
- *Mesh Parameters* (managing the splitting of individual edges and faces).
- *1D Collectors* (information on 1D element sets and their physical properties).
- *2D Collectors* (information on 2D element sets and their physical properties).
- *3D Collectors* (information on 3D element sets and their physical properties).
- *Connection Collectors* (information on connections of meshes and their properties).
- *CSYS* (list of local coordinate systems).

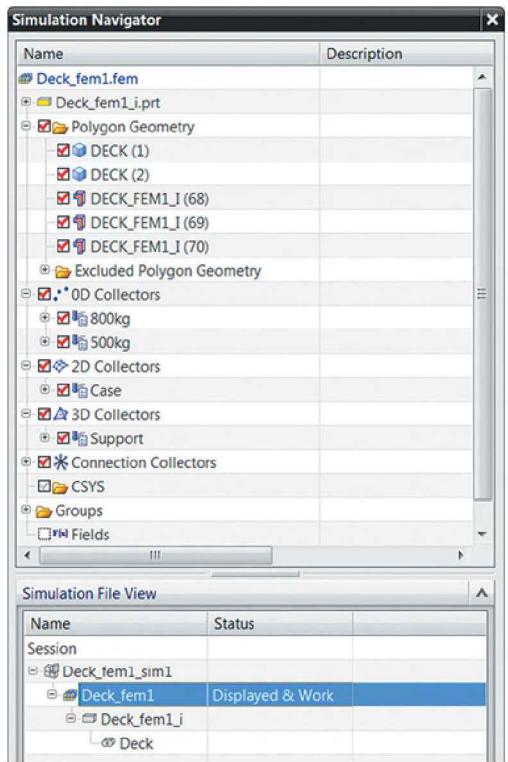


Figure 3.2. *Simulation Navigator* toolbar

You can use the **Simulation File View** dialog box of the **Simulation Navigator** toolbar to switch quickly between the master model, the idealized model, the FE model, and the simulation model.

You can work with all FE creation features using the **Simulation Navigator** of the active FE model. To use a command, choose it in the shortcut menu that appears when you right-click a node (Figure 3.3).

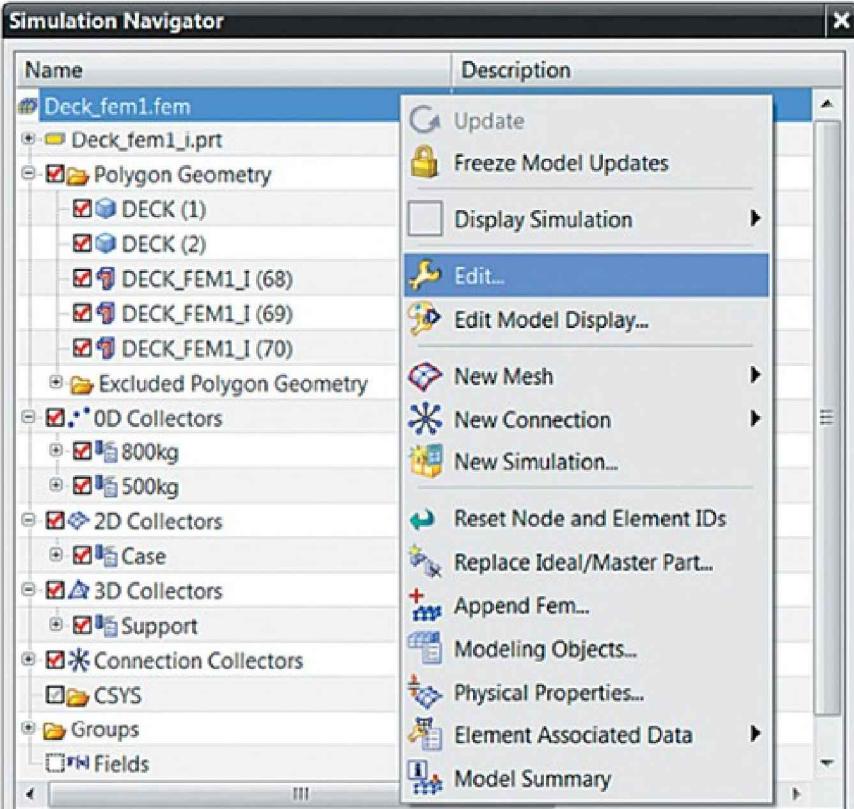


Figure 3.3. Starting a command from the Simulation Navigator toolbar

The FE model can contain a large number of finite element sets with different physical properties and material properties. There are three model tree views to help you check and validate the FE model data input: the *standard view*, the *material view*, and the *physical properties view*. Figures 3.2 and 3.3 show the *standard view* (default). To switch to *material view* or *physical properties view*, right-click in an empty model tree area and choose the view in the shortcut menu (Figure 3.4).

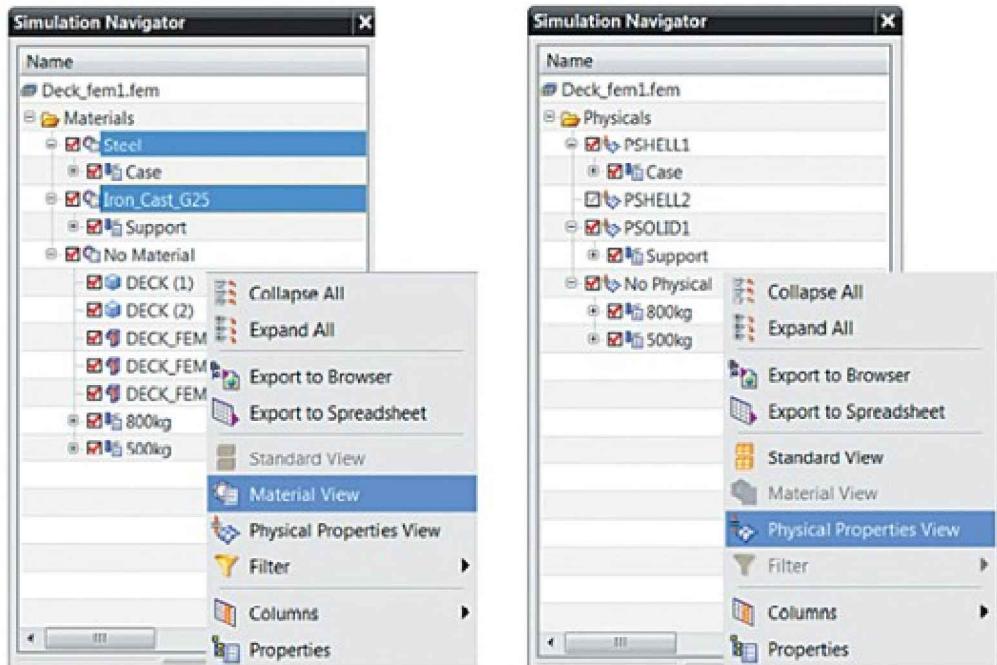


Figure 3.4. FE model tree views

3.2. Collector fundamentals. Materials

3.2.1. Working with collectors, physical properties

Each finite element of the FE model must have physical properties defined. In **NX Advanced Simulation** information of this type is organized in so-called collectors that contain tables of physical properties specified for individual element sets. An FE mesh collector is an element in the simulation model tree that contains information concerning the type of elements, their properties (physical property tables), display options, and simulation FE mesh parameters (Figure 3.5). There are several types of collectors, which mostly differ in dimensionality and topology of the finite element. Here is a brief description of these types in the order they appear in the model tree:

- **0D Collectors** are sets of elements such as point masses, elastic damper links and spot welds.
- **1D Collectors** are sets of one-dimensional elements such as beam, elastic damper, and other similar elements.
- **2D Collectors** are sets of spatial shell and flat elements.
- **3D Collectors** are sets of 3D solid-body elements.

- Connection Collectors are sets of mating conditions for meshes and contact meshes.

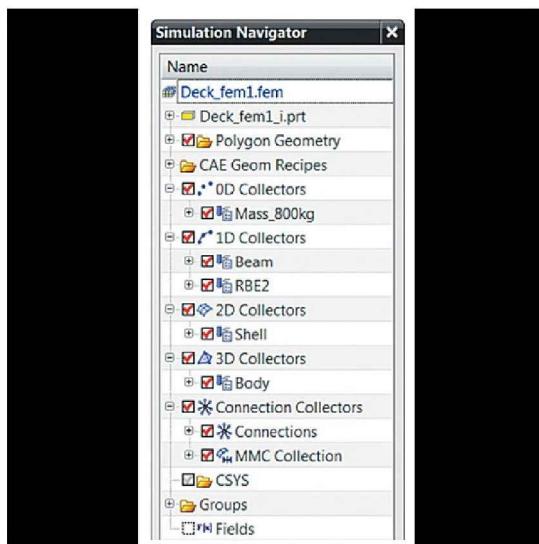


Figure 3.5. Types of collectors

A collector can contain several sets of finite elements that are in turn organized in physical property tables (Figure 3.6). If necessary, you can redefine parameters of element sets within a single collector type by physical property. You can do this by dragging the element set to the desired corner of the physical property table.

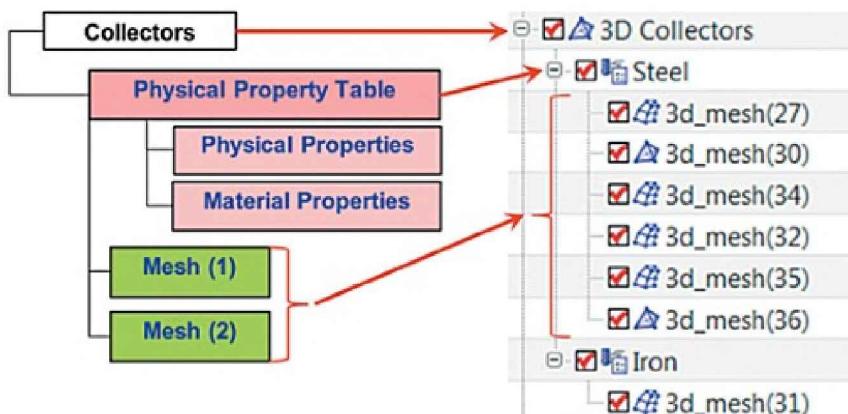


Figure 3.6. Populating a collector

You can use one of three workflows to create collectors when you design a FE model:

1. Creating a collector and a physical property table before you generate the mesh. This explicit workflow is useful for building highly complex models that contain several bodies, materials, and FE meshes. You can use this workflow to define model properties in a neat way and reduce the risk of simulation or calculation error.
2. Creating a collector and a physical property table at the time of mesh generation.
3. Automatic creation of a collector at the time of mesh generation with subsequent editing of the physical property table. This workflow allows to create simulation mesh files and FE model files with all necessary collectors quickly and automatically. It also allows inheriting object properties from the geometry CAD model or using default property values and parameters.

You can start the collector creation command using the following:

- Choose **Insert→Mesh Collector** in the main menu.
- Click the command on the **Advanced Simulation** toolbar.
- Right-click a collector node in the model tree on the **Simulation Navigator** toolbar and choose *New Collector*.
- Use the *Destination Collector* group in the 0D, 1D, 2D, or 3D mesh creation dialog box.

The **Mesh Collector** dialog box contains the following parameters (Figure 3.7):

- *Element Family*. Select an element type (0D, 1D, 2D, 3D, 1D/2D contact).
- Depending on the selected element family, specify the *Collector Type*, for example, for a 1D family you can specify one of the one-dimensional finite element types.
- In the *Properties* group, select a *Type* (in the case of 2D elements you can choose thin shell or multi-layer shell) and proceed to create *Solid properties* (for 3D family) or select existing properties from the list. To create new body properties, open the dialog box as shown in Figure 3.7. The content of this window changes depending on the families and types of elements selected.

Note that the names of the collector and the properties of the body can be

different. You can use default or custom names. In the standard tree view of the **Simulation Navigator** the nodes are sorted by collector names.

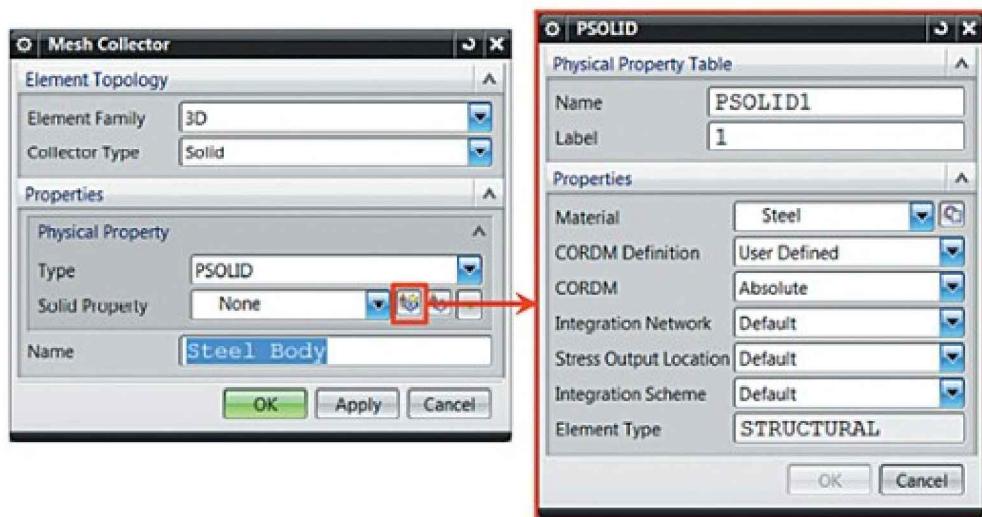


Figure 3.7. Creating a collector

In addition to primary properties of elements you can use *Mesh Associated Data* to specify extra parameters for individual sets of elements:

- Material Orientation
- Cross-section orientation of rod elements
- Elastic damper properties
- Inertial characteristics
- Shell element offset

3.2.2. Material Library

To perform any type of structure behavior analysis, you need to specify physical and mechanical properties of materials at the finite-element model creation step. **NX Advanced Simulation** provides two methods of specifying materials for finite elements:

1. Use the material selected for geometry bodies. In this case you can specify the material in the **Modeling** application when you process the master model (Master Part) or the idealized model (Idealized Part) in **Modeling** or **Advanced Simulation**. You also specify material parameters for polygon bodies in the FE model (FEM Part). The material assigned to the geometry is used as the default material for elements and

corresponds to the *Inherited* option in the materials list of the physical property table dialog box (Figure 3.8) .

2. Use the material specified in the collector. When you specify physical properties in the corresponding collector (Figure 3.8), select a material that exists in the FE model from the drop-down list (blue frame) or create a new material from the Materials Library (red frame).

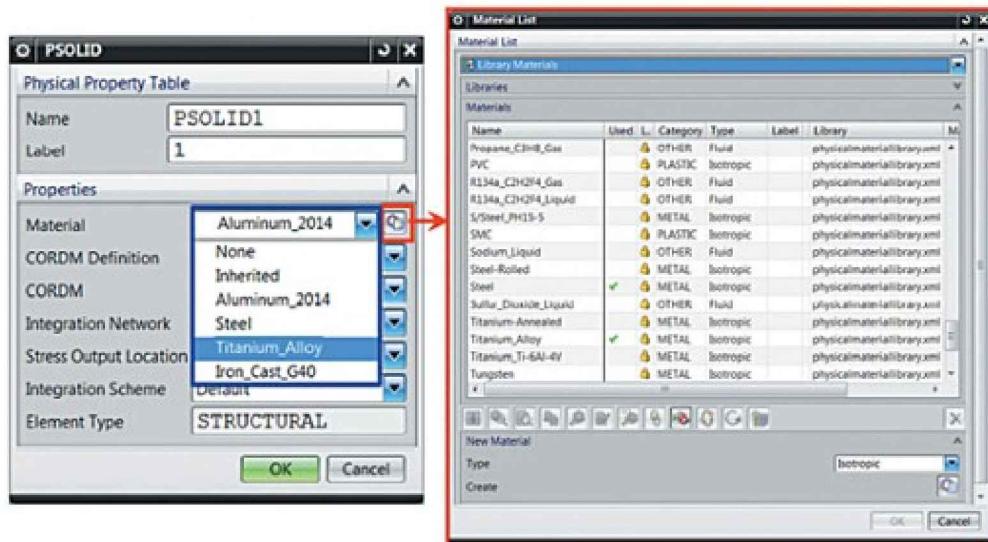


Figure 3.8. Assigning a material in physical properties

You can assign a material using the **NX Advanced Simulation Materials Library** or create a new material that will be saved only in the current FE model. Depending on the analysis type, you can use the following material types: isotropic, orthotropic, anisotropic, fluid, and several models of nonlinear materials (hyperelastic, shape memory, gasket-type materials and others.).

You can find material and library management tools (Figure 3.9) on the **Advanced Simulation** toolbar or in the main menu (**Tools→Materials**).



Figure 3.9. Material management tools

Use the **Assign Materials** command to assign materials to bodies. You can select the materials from either the built-in NX library, which contains standard material properties (including temperature-dependent ones), or a user-created library. Using this command involves the following basic steps (Figure 3.10):

1. Select one option from the **Type** list:
 - To select one or more bodies, use *Select Body(s)*.
 - To select all bodies in the model that have no material assigned, select *All Body(s) Without Assigned Material*.
 - To select all bodies in the model, select *All Bodies*.
2. In the graphics area, select the bodies to which you want to assign a material (if you use the *Select Bodies* selection type).
3. Select either *Library Materials* to choose one of the materials libraries, or *Local Materials* to use materials created in the FE model.
4. Select a material from the list.
5. Click **Apply**. A green check mark appears next to the relevant material in the *Used* column.

To view the details of material properties or to delete a material-to-body assignment (the material itself is not deleted), use commands in the bottom part of the dialog box.

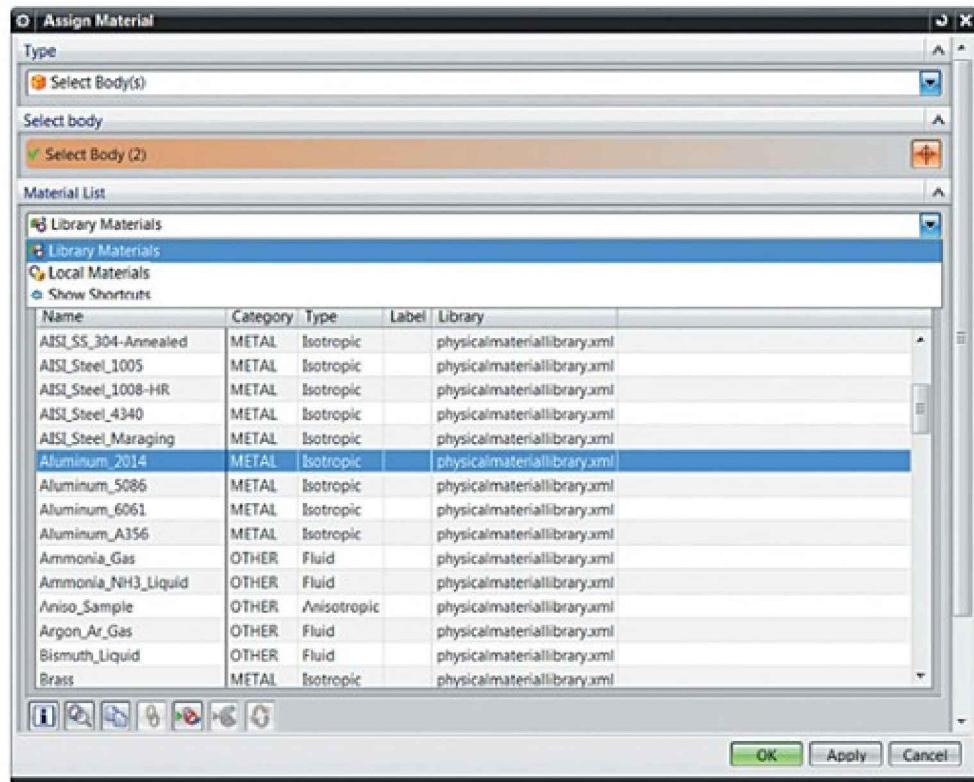


Figure 3.10. Assign Materials dialog box

Use **Manage Materials** to edit existing materials or create new isotropic, orthotropic, anisotropic, fluid, hyperelastic and other materials. The dialog bar is similar to the one for the **Assign Materials** command. To create a new material, do the following:

1. Select a material type from the list in the *New Material* group.
2. Click *Create Material*.
3. In the dialog box, type a unique *Name* for the new material.

Enter the material's physical and mechanical properties in the corresponding tabs and boxes. When you create constants, select *Expression*, when you create properties dependent on other physical values, select *Field*. Make sure you select correct units of measurement for each value you enter (Figure 3.11).

4. Click **OK**. A new material appears in the list.
5. Click **Close**.

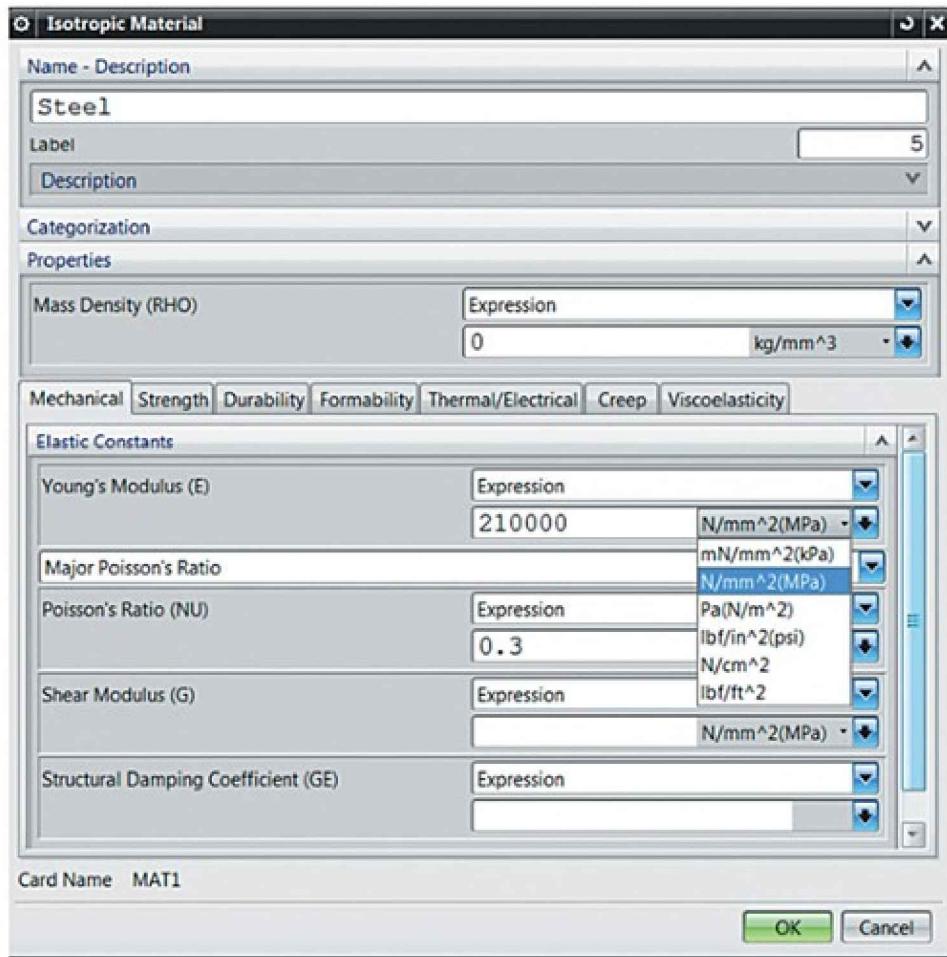


Figure 3.11. Material editing dialog box

You can use the **Manage Library Materials** command to edit an existing materials library or create a new one. You can select one of the following actions in the *Type* list of the dialog box: export a material to a library, modify a materials library, or delete a material from a library (3.12). Specify the location and files of materials libraries, select materials, and click **Apply** or **OK**.

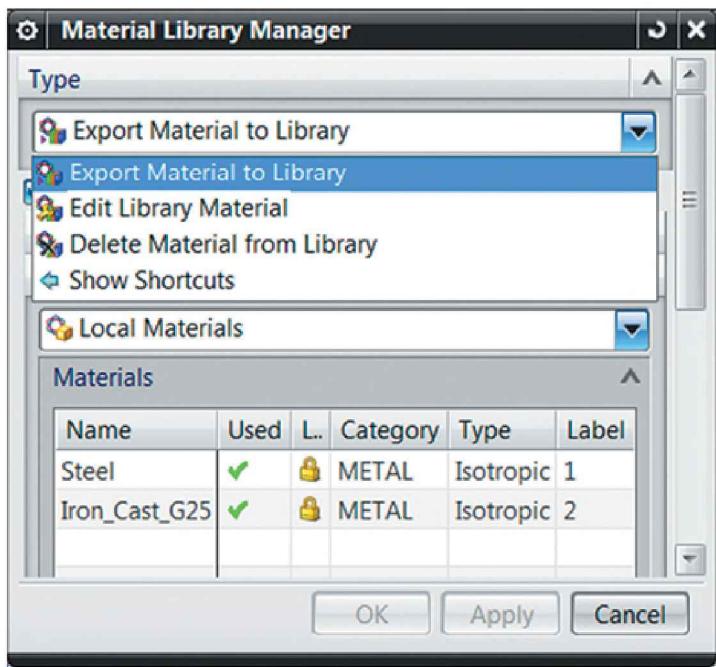


Figure 3.12. Materials library management dialog box

3.3. Creating a finite-element model

Building the FE simulation mesh involves discretizing the mathematical model, that is, dividing the continuous geometry structure into finite elements. Each element is a mathematical representation of a discrete part of the entire structure being described. The element has an inherent interpolation function. Creating the finite-element mesh is one of the most important and critical stages of numerical engineering analysis, and the accuracy of results directly depends on the quality of the simulation mesh you create.

You can use the FE mesh creation capabilities of NX Advanced Simulation to create:

- 0D: scalar elements at selected points.
- 1D: one-dimensional elements on edges or lines.
- 2D: two-dimensional elements on faces and surfaces.
- 3D: solid-body elements in volumes.

The system saves all parts of the FE mesh and the relevant parameters

such as material, thicknesses, sections (element attributes) in an FEM file. You can create an FE mesh using geometry created in other NX applications or imported from other CAD systems.

You can access FE mesh creation commands using one of several methods (Figure 3.13):

- Choose **Insert→Node/Element/Mesh** in the main menu.
 - Click the command on the **Advanced Simulation** toolbar.
 - Use commands on the **Finite Element Model** toolbar.
 - Work with nodes and elements using commands on **Node Operations** and **Element Operations** toolbars.
 - Use the model tree view of **Simulation Navigator** by right-clicking a 3D/2D/1D/0D collector node and choosing **New 3D/2D/1D/0D mesh** in the shortcut menu.

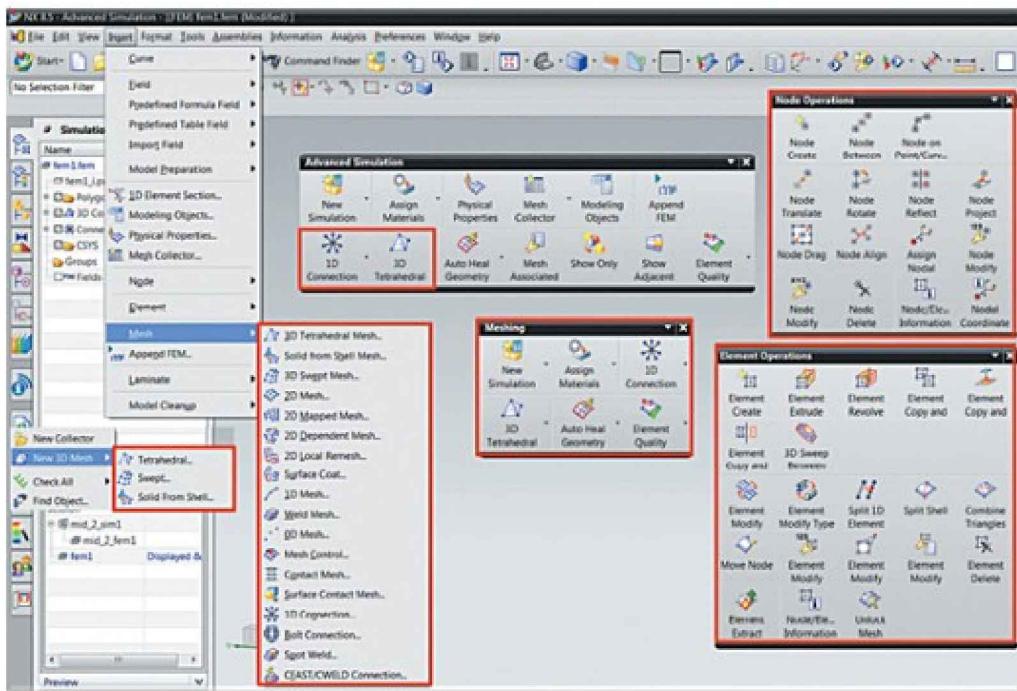


Figure 3.13. FE mesh creation tools

3.3.1. Creating 3D meshes

Spatial solid-body finite elements are used for modelling of thick plates and solid bodies. For mechanical cases each node of a 3D element has three translational degrees of freedom. Before creating a mesh in the body using

spatial solid-body elements, you need to choose the type of finite elements. Consider a list of the most important 3D elements for the NX Nastran solver.

Element name	Description	Type of physical properties
CHEXA(8)	8-node hexahedral solid-body element	PSOLID PLSOLID (hyperelastic)
CHEXA(20)	20-node hexahedral solid-body element	PSOLID PLSOLID (hyperelastic)
CPYRAM	5-node or 13-node pyramid element	PSOLID
CTETRA(4)	4-node tetrahedral solid-body element	PSOLID PLSOLID (hyperelastic)
CTETRA(10)	10-node tetrahedral solid-body element	PSOLID PLSOLID (hyperelastic)

You can generate 3D FE meshes in **NX Advanced Simulation** using commands that can be used separately or in combination:

- Use **3D Tetrahedral** to create a mesh with CTETRA4 and CTETRA10 tetrahedral finite elements and pyramidal finite elements containing from 5–13 nodes (CPYRAM). For a detailed description, see below.
- Use **Solid From Shell Mesh** to create a 3D tetrahedral mesh in a closed region with boundaries previously divided into auxiliary 2D triangular finite elements. This command is useful when you don't have a polygon body but you have element faces fully enclosing a region of space.
- Use **3D Swept Mesh** to create a structured 3D hexahedral mesh using CHEXA8 and CHEXA20 elements. For a detailed description, see below.

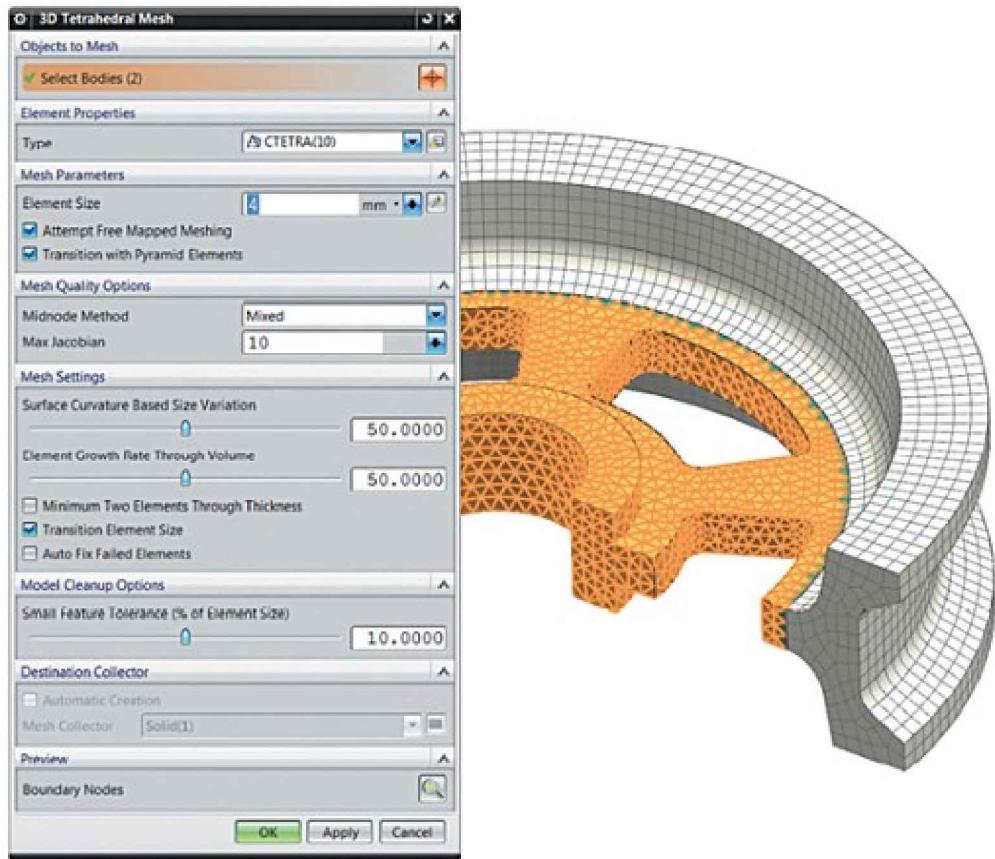


Figure 3.14. Creating a tetrahedral mesh

To create 3D tetrahedral finite elements in selected bodies, open the **3D Tetrahedral** dialog box and specify mandatory options (Figure 3.14):

- Use *Select Bodies* to specify bodies that are not divided into elements.
- Select *Type (Element Properties)* to specify the element type, for example quadratic tetrahedral *CTETRA(10)*.
- Enter *Element Size* to specify the characteristic size of the elements in the selected length unit.
- In the *Destination Collector* group, select an existing collector from the list (select *Automatic Creation* check box) or create a *New Collector* in the same dialog box.

You can use the following additional options to control the parameters of the FE mesh that you generate:

- Select *Automatic Element Size* to automatically calculate the element size for each body, taking into account its topology and outline dimensions.
- Select *Attempt Free Mapped Meshing* to try and create a regular triangular mesh on faces of the selected body.
- Select *Transition with Pyramid Elements* to create pyramid elements to ensure correct transition from hexahedral to tetrahedral elements. This option is only available if you create a tetrahedral mesh in a volume adjacent to a volume discretized with a hexahedral mesh.
- Use the *Mesh Quality Options* group to control the projection of intermediate nodes of quadratic tetrahedral elements onto source geometry.
- Use the *Mesh Settings* and *Model Cleanup Options* groups to specify additional mesh generation options:
 - *Surface Curvature Based Size Variation*: the name is self-explanatory; the smaller the radius of curvature, the more finite elements are used to describe it. This option controls the degree of description of curvilinear surfaces by elements. Figure 3.15 shows some command results with this option set to different values.

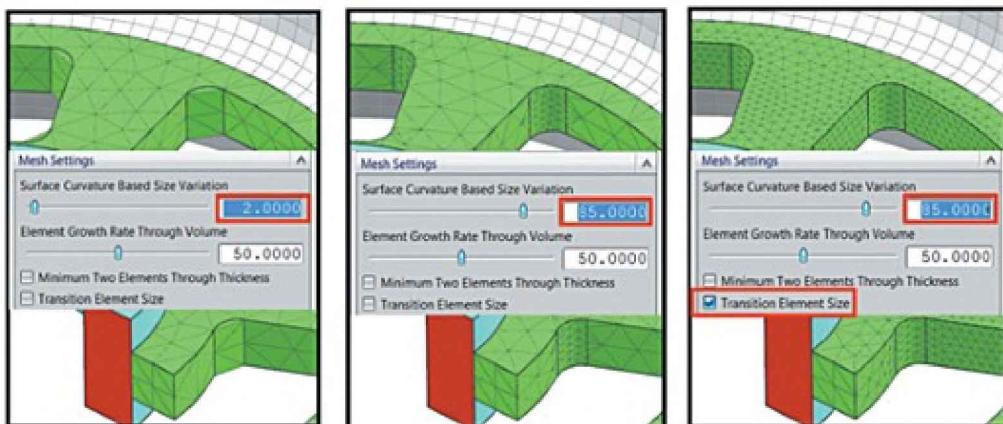


Figure 3.15. Element size based on curvature

- *Element Growth Rate through Volume*: control the degree of discretization within the selected body relative to the size of elements on its boundary.

- *Transition Element Size*: use this option to enable gradual transition from the local element size at the boundary, specified for example by the **Mesh Control** command, to the general size of elements for the selected body (Figure 3.16).

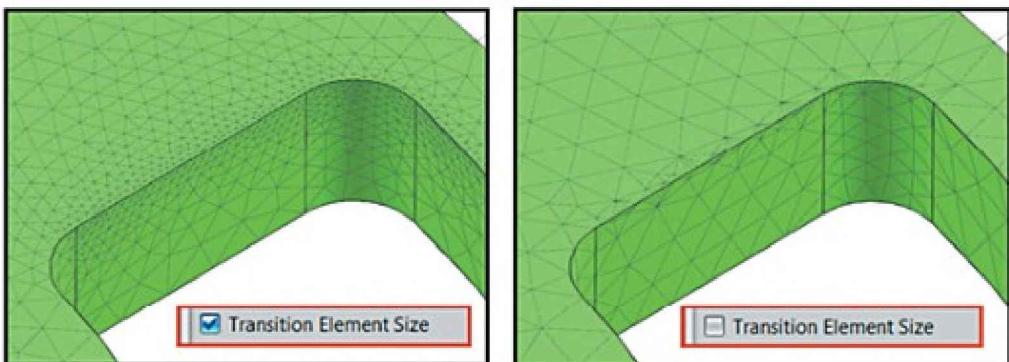


Figure 3.16. Element size based on curvature

- *Auto Fix Failed Elements*: automatically recreate the mesh whenever elements not conforming to quality criteria are discovered.
- *Small Feature Tolerance*: control the acceptable characteristic size of geometric features that are ignored when creating the mesh.
- If you use *Preview*, you can see nodes that will be created at body boundaries. If quantity or position of these nodes is not satisfactory, set a different element size or modify other options.

For most bodies it is advised that you create a mesh that contains only hexahedral elements. However, an automatic hexahedral element meshing is not possible for most geometries, so the meshing is done on subregions. To prepare the geometry for the necessary splitting, use the body splitting commands for the idealized or master model (Chapter 2, part 1). Also keep in mind the abstraction of polygon geometry within the finite-element model (para. 3.6). Before we proceed to describe the structured hexahedral mesh creation command, note the most important rules and special aspects of generating such meshes.

You can use the **3D Swept Mesh** command to create a regular mesh that consists of hexahedral and prismatic elements by stretching the faces of flat

elements from one side of the body (source faces) to the other (target face). The body is filled with elements layer-by-layer (Figure 3.17). To control the quality of the FE mesh in the “cross-section” of the sweep body, you should do the following:

- Create a template 2D mesh on source faces before starting the operation.
- Use options of the **3D Swept Mesh** command to control the creation of the mesh on source faces during hexahedral element generation.

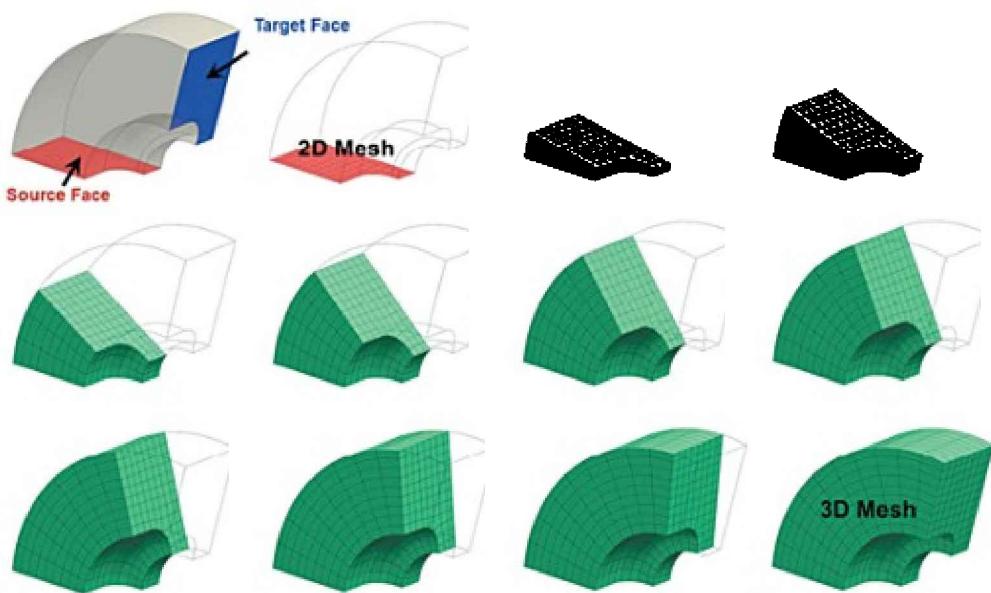


Figure 3.17. Generation of a 3D hexahedral mesh layer-by-layer

Observe the following rules when you create a 3D hexahedral mesh:

- You can specify several source faces but only one target face. If necessary, you can use **Merge Face** to prepare a single target face.
- The source and target faces may not have shared edges or vertices.
- You must be able to create a regular mesh on generating faces (all faces of the body except the source and target ones).
- The generating faces may not be parallel to source and target faces.

- Source and target faces can be flat or curvilinear.

You can create hexahedral meshes in NX Advanced Simulation in two ways:

- Automated (*Multi Body-Infer Target*), which creates hexahedral meshes for several bodies at once if you select one source face for each body (Figure 3.18). If you use this method, the source and target faces of each body must have identical topology and contain the same number of edges.
- Manual (*Until Target*), which unlike the automated method, allows to generate a mesh for only a single body but also allows to select several source faces instead of just one (Figure 3.19). This method is used for bodies with complex shapes and several source faces.

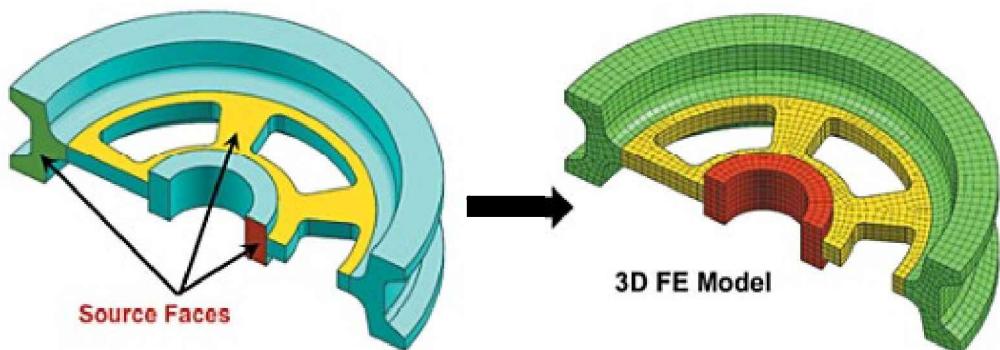


Figure 3.18. Automated generation of a hexahedral mesh

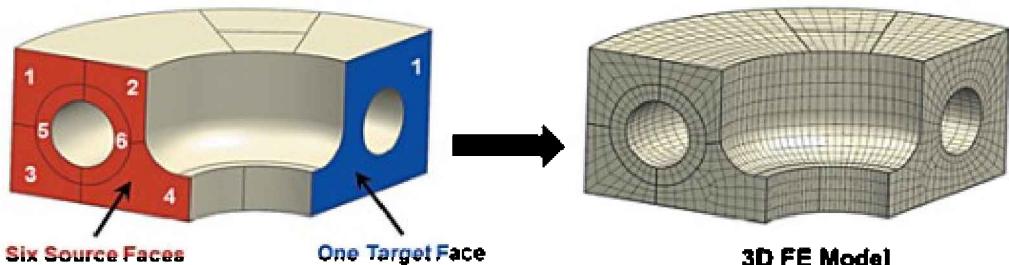


Figure 3.19. “Manual” generation of a hexahedral mesh

To create 3D hexahedral elements, open the **3D Swept Mesh** dialog box and specify mandatory parameters (Figure 3.20):

- In the *Type* group, select the mesh generation method.

- In the *Objects to Mesh* group, if *Multi Body-Infer Target* method is selected, select the *Select Source Face* option, then select a source face for each of the split bodies. If *Until Target* method is selected, select one or several source face of the body (*Select Source Face*) and one target face of the body (*Select Target Face*).
- Select *Type (Element Properties)* to specify element type, for example linear hexahedral element CHEXA(8).
- Enter the characteristic element size in the *Source Element Size* box.
- In the *Destination Collector* group, select an existing collector from the list, opt to create a collector automatically (select the *Automatic Creation* check box), or create a new collector.

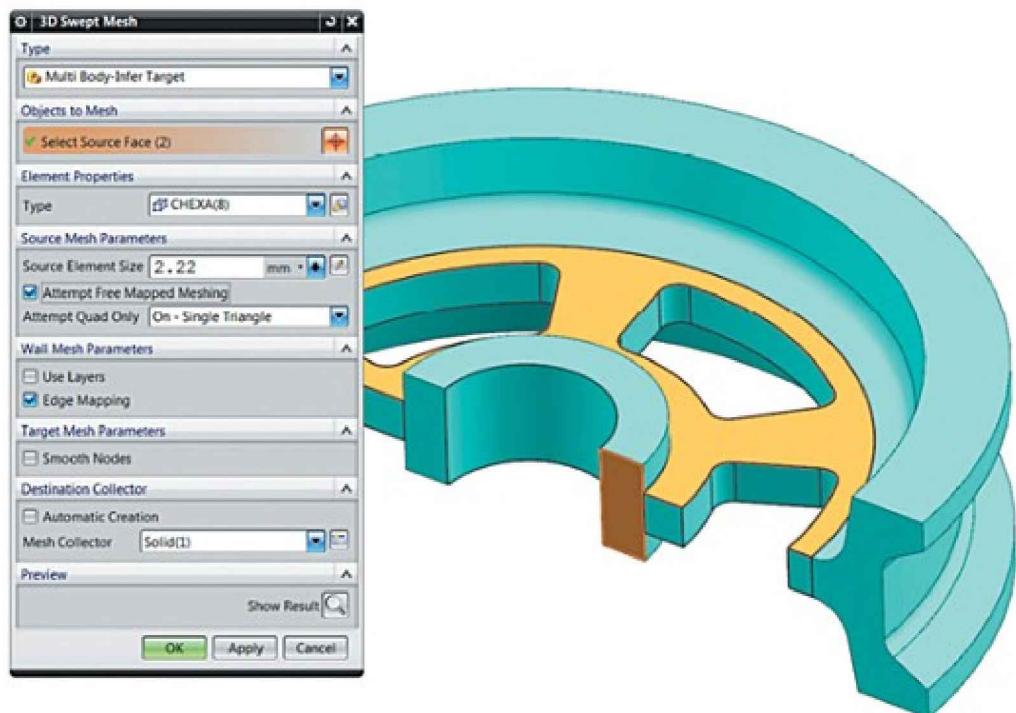


Figure 3.20. Creating a hexahedral mesh

In a similar way to the tetrahedral FE mesh creation command, you can use some extra parameters of the **3D Swept Mesh** command:

- Select *Attempt Free Mapped Meshing* to try and create a regular quadrangular mesh on faces of the selected body.

- Set *Attempt Quad Only* to restrict the usage of triangular elements on the source face: allow triangular elements, allow only quadrangular elements, or create only one triangular element.
- Select *Edge Mapping* to enable exact projection of nodes from source face edges to target face edges (Figure 3.21).

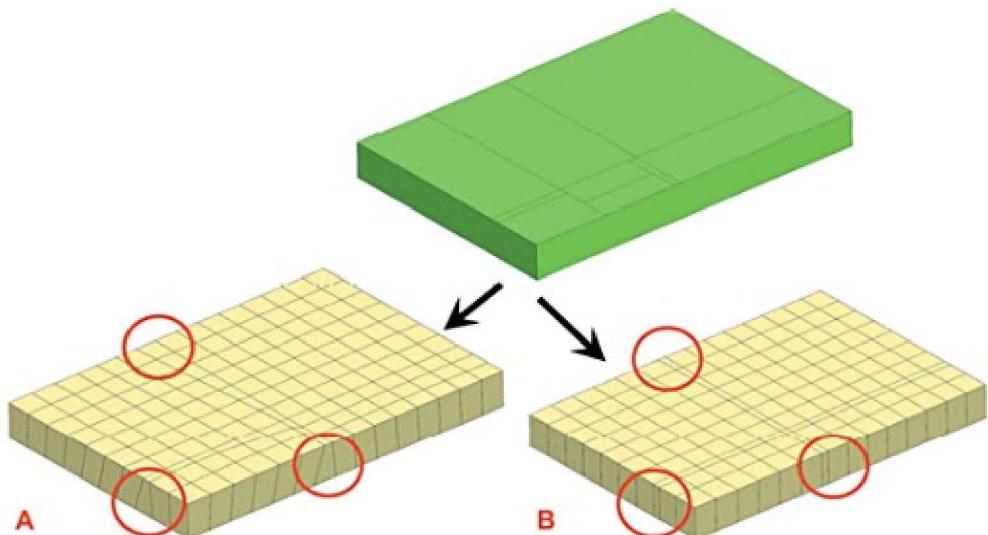


Figure 3.21. Edge Mapping:

A. Option not selected. B. Option is selected

- Select *Use Layers* to control the number of element layers created on generating faces. If this option is not selected, the number of layers depends on the specified characteristic element size.
- Select *Smooth Nodes* to redistribute nodes within the internal region of the target face to achieve a more uniform mesh compared to the source face mesh. This option is effective if there are several source faces (Figure 3.22).

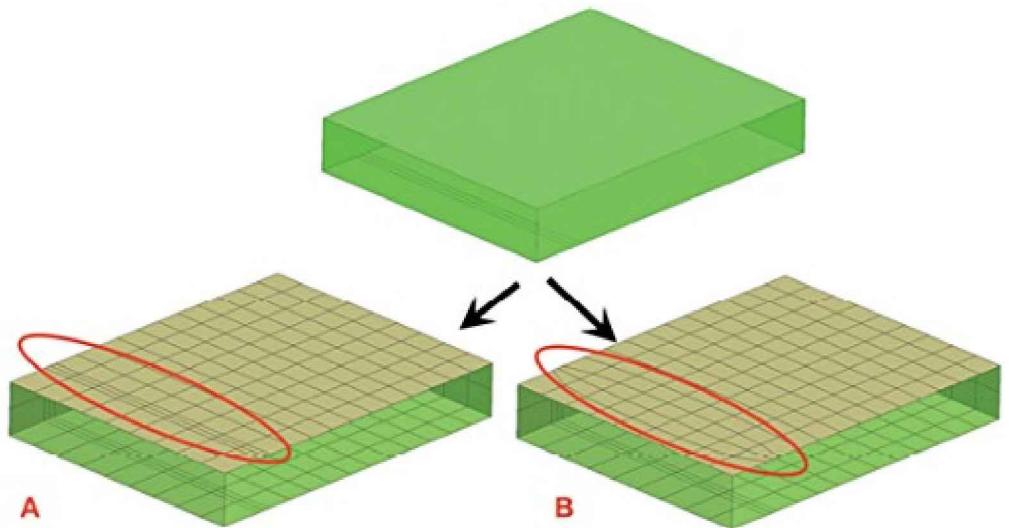


Figure 3.22. Smooth Nodes:

A. Option not selected. B. Option is selected

- If you use *Preview*, you can see the nodes of the boundaries of the body that will be created when the current command is selected. If the number or arrangement of nodes is unsatisfactory, enter a different element size or change other creation parameters.

3.3.2. Creating 2D meshes

If you model structures whose thickness is small compared to other dimensions, you can reduce the dimensionality of the problem by using so-called two dimensional [2D] elements instead of three-dimensional solid elements. These elements are typically used to model shells and thin plates. Here a list of the most important 2D elements of the NX Nastran solver.

Element name	Description	Type of physical properties
CQUAD4	4-node quad shell element	PSHELL, PLPLANE, PCOMP, Lamine
CQUAD8	8-node quad shell element	PSHELL, PLPLANE, PCOMP, Lamine
	4-node quad shell element with stiffness of	PSHELL, PCOMP,

CQUADR	rotational degree of freedom normal to the surface	Laminate
CQUADX4	4-node quad axisymmetric element	PSOLID, PLSOLID
CQUADX8	8-node quad axisymmetric element	PSOLID, PLSOLID
CSHEAR	4-node quad shear panel element (resists only shear forces in the element plane)	PSHEAR
CTRAX3	3-node triangular axisymmetric element	PSOLID, PLSOLID
CTRAX6	6-node triangular axisymmetric element	PSOLID, PLSOLID
CTRIA3	3-node triangular shell element	PSHELL, PLPLANE, PCOMP, Laminate
CTRIA6	6-node triangular shell element	PSHELL, PLPLANE, PCOMP, Laminate
CTRIAR	3-node triangular shell element with stiffness of rotational degree of freedom normal to the surface	PSHELL, PCOMP, Laminate

Another possible application of 2D elements is creating an auxiliary template mesh and using the 2D mesh to subsequently create 3D meshes. You do not have to specify any physical properties for auxiliary 2D elements because these elements are not part of the solution.

You can generate a 2D FE mesh in **NX Advanced Simulation** using the following commands:

- Use the **2D Mesh** command to create a so-called freeform (irregular) mesh with triangular and quadrangular elements.
- Use the **2D Mapped Mesh** command to create a 2D fully regular mesh for simply connected geometry regions.
- Use the **2D Dependend Mesh** command to create several 2D meshes dependent on a single 2D mesh. You can use this command to create identical meshes relative to a single master face. If you modify the master face mesh, meshes on dependent faces are updated automatically.
- Use the **Surface Coat** command to generate 2D elements based on existing 3D element faces.

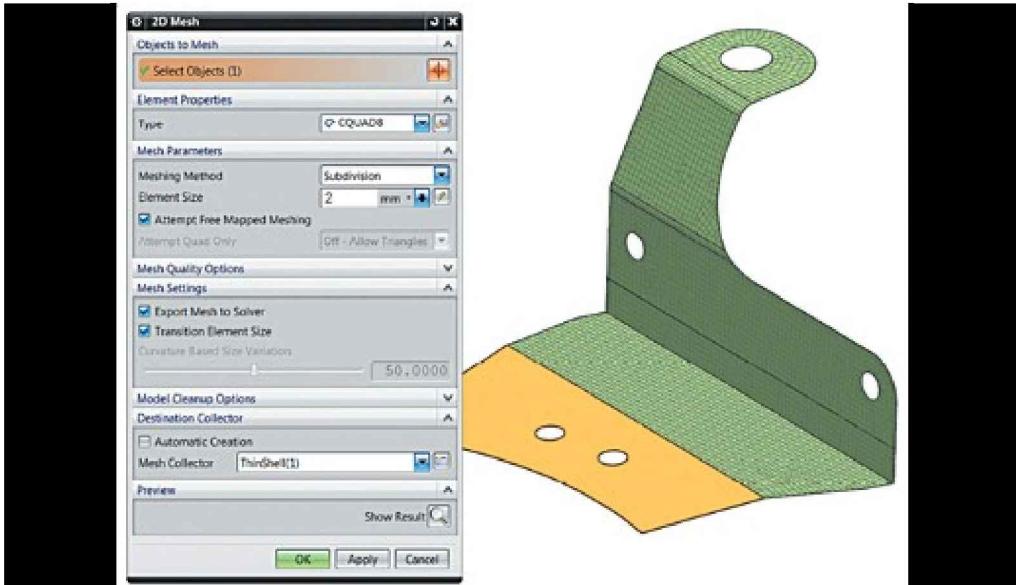
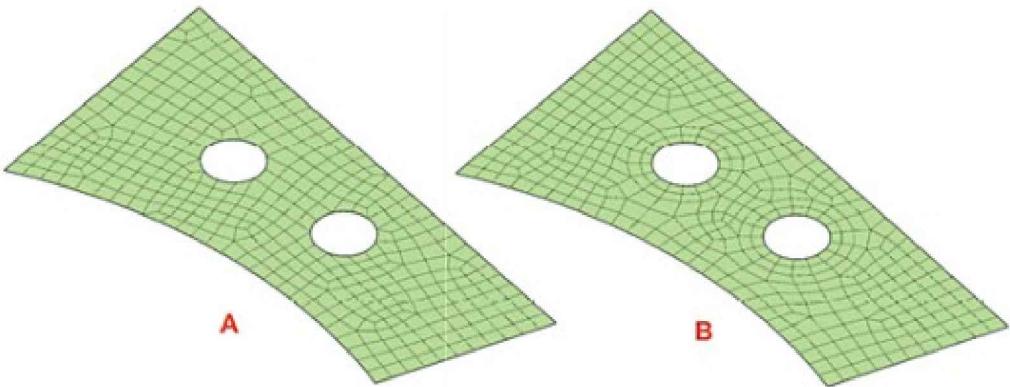


Figure 3.23. Creating a 2D freeform mesh

You can create a 2D irregular mesh on the specified faces in the **2D Mesh** dialog box by specifying mandatory parameters (Figure 3.23):

- Use *Select Objects* to specify faces that are not split into elements.
- Select *Type (Element Properties)* to specify the element type, for example quadrangular CQUAD8 with midside nodes.
- Select the *Meshing Method* that is used for splitting faces into finite elements. You can use the *Subdivision* method to create a reasonably uniform mesh of elements on faces with the specified element size. You can use the *Paver* method to create a regular mesh near free boundaries of the selected face (Figure 3.24). In this way you can describe holes and similar geometry more precisely.



*Figure 3.24. Meshing Method:
A. Subdivision method. B. Paver method*

- Enter *Element Size* to specify the characteristic element size in the selected length unit.
- In the *Mesh Settings* group, you can specify whether the newly created elements should be used as auxiliary (clear the *Export Mesh to Solver* check box) or as physically meaningful elements used in the simulation. For example, if you use shell or plate elements to simulate thin-shell structures, you should select the *Export Mesh to Solver* check box.
- Select *Transition Element Size* to gradually transition from the element size used locally at the boundaries to the general element size.
- Use the *Destination Collector* group to select the collector that contains the physical properties of elements. You can specify an existing collector from the list (clear the *Automatic Creation* check box), or you can specify automatic collector creation (select the *Automatic Creation* check box). Alternatively, you can create a new collector right in the current dialog box.
- If you use *Preview*, then on the boundaries of the body you can see the nodes that will be created when the current command is executed.

The **2D Mesh** command takes several more special parameters:

- If you clear the *Attempt Free Mapped Meshing* check box, the *Attempt Quad Only* option becomes available. This option regulates the usage of triangular elements on faces.

- If you select the *Split Quad* check box, the mesh generator monitors the quality of quadrangular elements that it creates, and a “bad” quadrangular element is split into two triangular ones if necessary.
- The *Model Cleanup Options* allow automatic removal or ignoring possible problem zones that occur due to imprecise geometry representation.

To create a fully regular 2D mesh on the specified faces using linear or quadratic, triangular or quad elements, you can use the **2D Mapped Mesh** command. The regularity of element arrangement on the face allows better control of their distribution compared to an irregular freeform mesh. In particular, you can fully control the quantity and quality of elements. From the practical point of view, using a regular mesh is preferable in areas with high gradients of values, for example in stress concentration areas around holes or blends. You can use the **2D Mapped Mesh** command to create regular meshes not only for quadrangular faces but also for triangular and some simply connected polyangular faces.

When you create regular meshes, including on polyangular faces, NX Advanced Simulation automatically finds corners which define the opposite sides of a region. Figure 3.25 shows examples of regular meshes with numbered corners. The *Edge Density* parameter is determined in accordance with the specified element size. This parameter controls the number of elements on each edge following these rules:

- For a quadrangular face, the number on opposite edges must be identical, as shown in Figure 3.25, A. Note the matching partitioning pattern on edges n1, and the n2 pair of edges.
- For a polyangular face, the sum of partitions on edges between the 1–4 pair of corners (partitions n2, n3, and n4) must be equal to the sum of partitions of edges for the opposite 2–3 pair of corners (partitions n2 + n3 + n4). Moreover, as shown in Figure 3.25, B, the partitioning of n1 edges must be identical.
- For a triangular face, the edges sharing a vertex n1 (where mesh degeneracy is expected) must be divided identically (Figure 3.25, C). Avoid creating a fully regular mesh on triangular faces because one of the vertices will have degenerate elements with angles

tending to zero as the number of elements increases. When you use the command to create a regular mesh for a triangular face, you should select the *Quad Only on 3 Sided Faces* option to generate a structural mesh (Figure 3.25, D). In this case you also need to ensure the division on edge n2 is even.

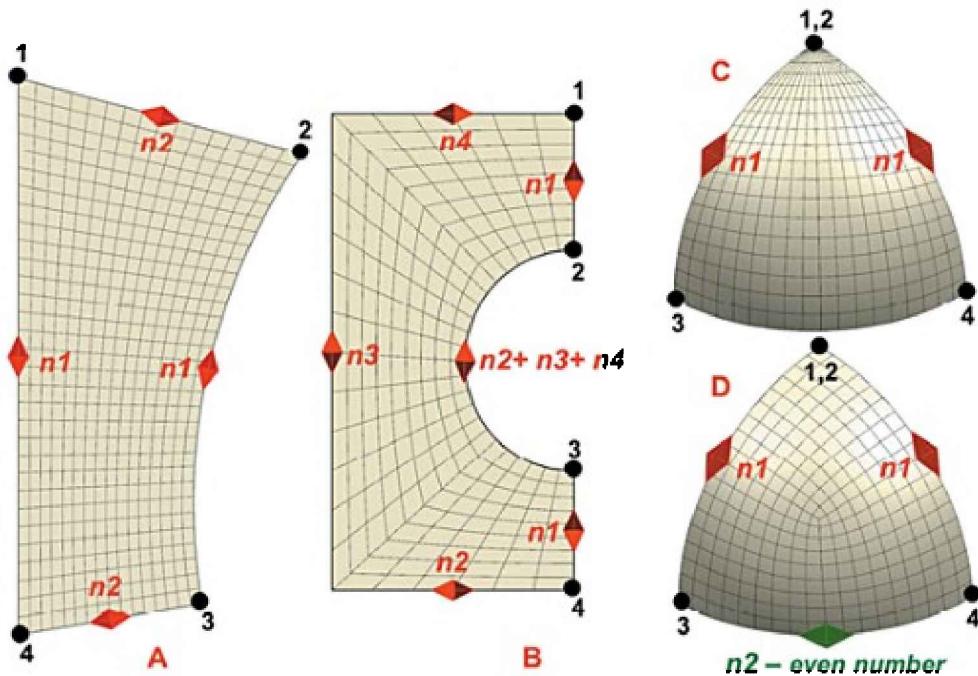


Figure 3.25. Examples of regular meshes:

A. Quadrangular region. B. Hexangular region. C, D. Triangular region

You can create a 2D regular mesh on the specified faces in the **2D Mapped Mesh** dialog box by specifying mandatory parameters (Figure 3.26):

- Select *Type (Element Properties)* to specify the type of 2D elements, for example quadrangular CQUAD4.
- Use the *Destination Collector* group to select the collector that contains the physical properties of elements. You can specify an existing collector from the list (clear the *Automatic Creation* check box) or specify automatic collector creation (select the *Automatic Creation* check box), or alternatively create a new collector right in the current dialog box.
- Under *Select Objects* select faces to create the regular mesh.
- Enter *Element Size* to specify the characteristic size of the element

for the selected unit of measurement.

- In the *Mesh Parameters* group, specify whether the newly created elements should be used as template elements or as physically meaningful elements participating in the simulation.
- After specifying the primary parameters for mesh generation you should preview the results. To do so, in the *Preview and Modify Mesh Constraints* group, click *Show Result*. The following options become available (Figure 3.26):
 - Under *Define Corners*, you can redefine corners that are used for finding the opposite sides of the region.
 - Under *Edge Density*, you can click a number of partitions symbol in the graphics area and specify a custom value in a dialog box. The mesh is automatically displayed on the screen, allowing to adjust the mesh parameters efficiently.

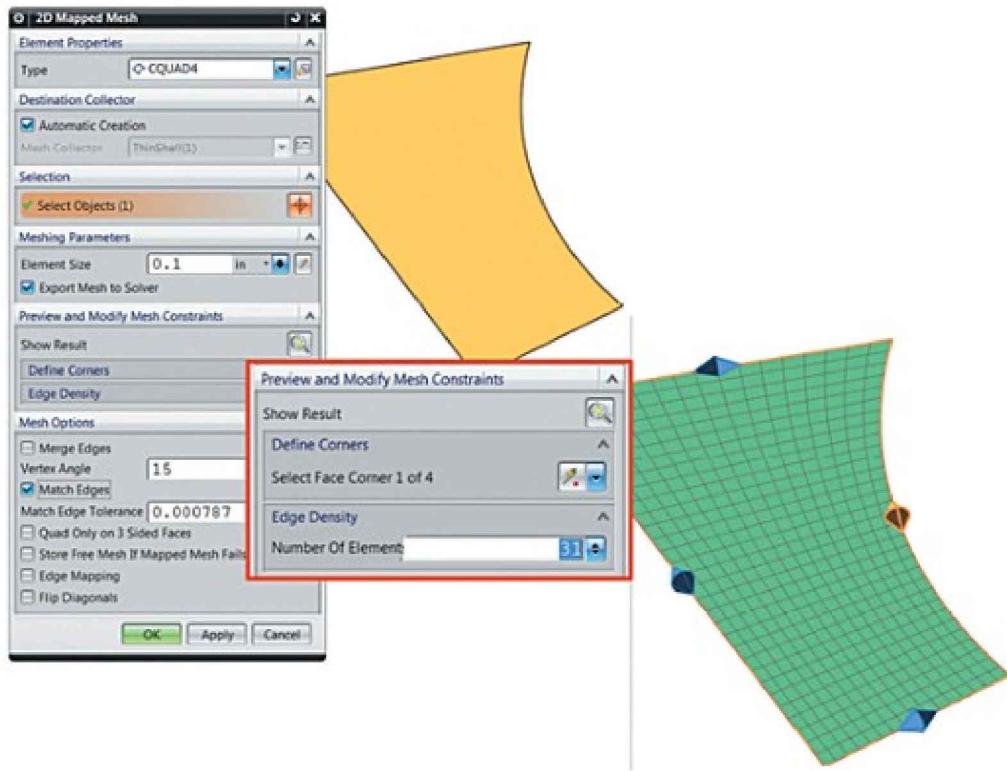


Figure 3.26. Creating a 2D regular mesh

To control the FE mesh generation more precisely, use additional options in the *Mesh Options* group of the **2D Mapped Mesh** command:

- Select *Merge Edges* to remove vertices of two or more edges if the angle between them is less than or equal to the value specified in the *Vertex Angle* box.
- Select *Match Edges* to generate a continuous regular mesh for several faces with the specified matching precision (*Match Edge Tolerance*).
- Select *Quad Only on 3 Sided Faces* to avoid element degeneracy in triangular regions (Figure 3.25, C). A structured mesh is created, which contains only quadrangular elements (Figure 3.25, D).
- If the current command cannot create a fully regular mesh on the selected face, you can create a freeform mesh (similar to the **2D Mesh**) command. To do so, select *Store Free Mesh if Mapped Mesh* fails.
- Select *Edge Mapping* to achieve a more regular and uniform mesh by projecting nodes from an edge to the opposite edge. The same option is applied when creating regular meshes on generating faces (Figure 3.21) when you use the **3D Swept Mesh** command.
- Select *Flip Diagonals* to flip the diagonals of triangular elements (Figure 3.27).

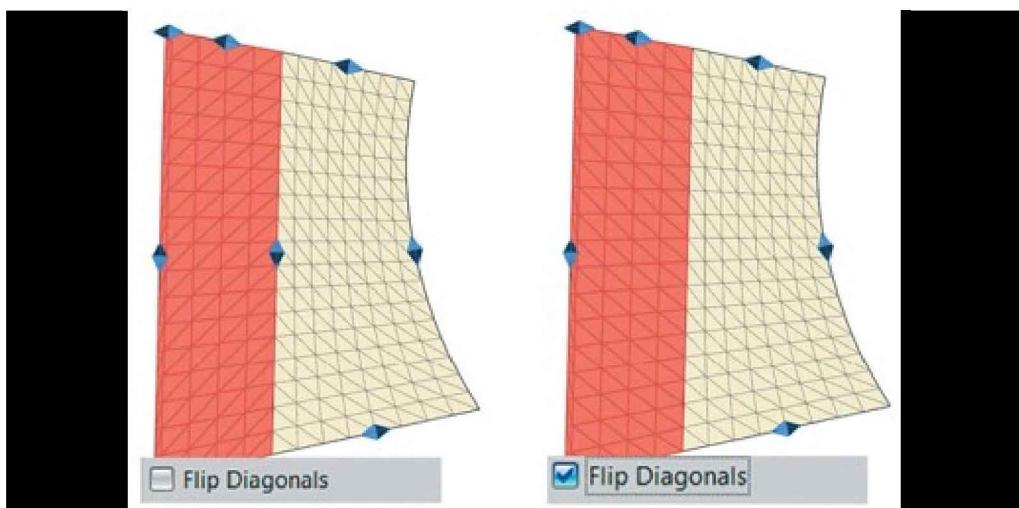


Figure 3.27. Flip Diagonals

For some finite-element simulations you need to create identical FE meshes on different faces. The previously described commands may not be sufficient to make sure the meshes are identical. In this case you should use

the **2D Dependent Mesh** command to create identical meshes on different faces. It creates a link between the master face mesh and the dependent face mesh. The dependent face mesh is rebuilt whenever the master face mesh is modified.

You can create two types of dependent meshes:

- **General:** identical meshes are created for geometrically similar faces, that is, faces with similar topology and the same number of edges, differing only in geometrical dimensions.
- **Symmetric:** identical meshes are created for completely identical faces. The nodes are created on the dependent face by translating these nodes from the source face relative to a defined coordinate system (Figure 3.28). This method is used to create identical meshes on matching faces of cyclically symmetrical models. The user can specify constraint equations on those faces which then can be used to set up cyclical symmetry boundary conditions.

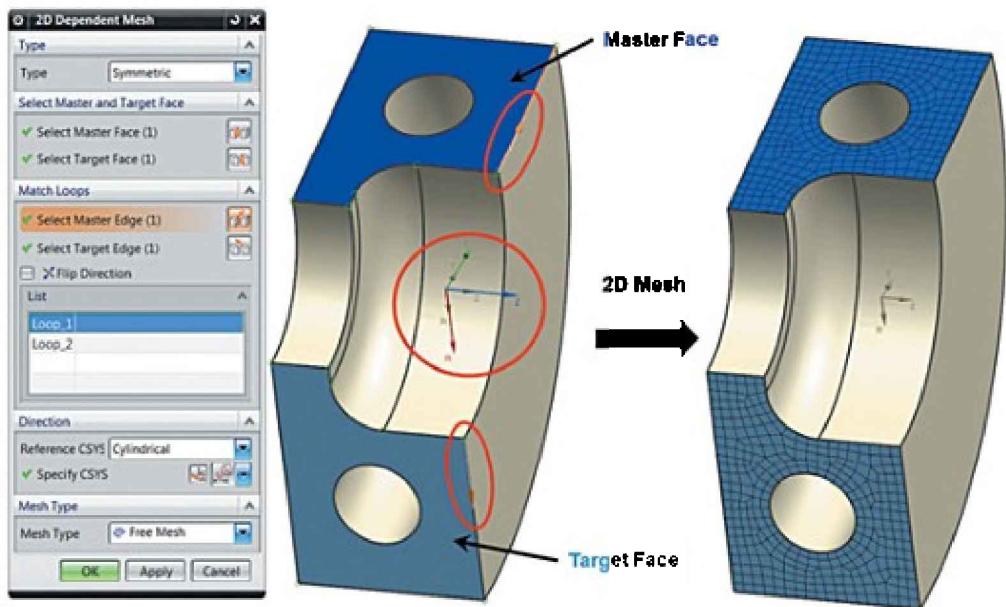


Figure 3.28. Creating a 2D dependent mesh

You can create a 2D dependent mesh on specific faces in the **2D Dependent Mesh** dialog box by setting up the following options (Figure 3.28):

- Select *Type* to specify the dependent mesh creation method (*General or Symmetric*).
- Under *Select Master Face*, specify the master face where the primary mesh is created.
- Under *Select Target Face*, specify the target face where the dependent mesh is created.
- In the *Match Loops* group, select pairs of similar edges (*Loop_1*, *Loop_2* and so on to ensure unambiguous correspondence of dependent faces. These edges, one edge per face, are added to the *List*. The number of edge pairs you specify depends on the face topology: for a simply connected region you select a single pair of edges, for a doubly connected region you select two pairs, and so on. Make sure you select the correct “traversal direction” of the edges. If necessary, you can flip the direction by selecting the relevant check box.
- In the *Direction* group, for the symmetrical method of dependent mesh creation, select or create a coordinate system in which the nodes are transferred from the master face to the dependent face.
- The final step involves specifying the type of the mesh (freeform or regular) you use to divide the master face and consequently, the dependent face (*Mesh Type* option).

After you confirm the command by clicking **OK** or **Apply**, a standard 2D mesh creation dialog box opens where you can specify all mesh creation options. To finalize the mesh creation on faces, click **OK**.

To minimize the degrees of freedom and therefore save time on simulating the layers of the structure, it can make sense to forgo 3D elements and use 2D shell elements instead if the geometry and boundary conditions permit. You can create such shell elements using the **Surface Coat** command, which allows to generate 2D meshes (Figure 3.29) based on the freeform 3D element faces you created. If you modify the geometry or FE mesh of the object that serves as the basis of the surface layer, the 2D mesh is automatically rebuilt as well. This layer modeling can be useful, for example, if you need to take into account additional mass, add stiffness, a cushion between bodies, and many other aspects.

To create a surface layer, open the **Surface Coat** dialog box and set the

following options:

- Select a Mode: *3D Mesh*, or *Element Face*, or *Geometry* as the basis of 2D elements.
- Depending on the specified mode, you can select objects of the corresponding type for surface mesh creation.
- Select *Type (Element Properties)* to specify the type of 2D elements, for example quadrangular CQUAD8.
- In the *Destination Collector* group, you can select a collector with physical properties of the elements, just like in other FE mesh building commands.

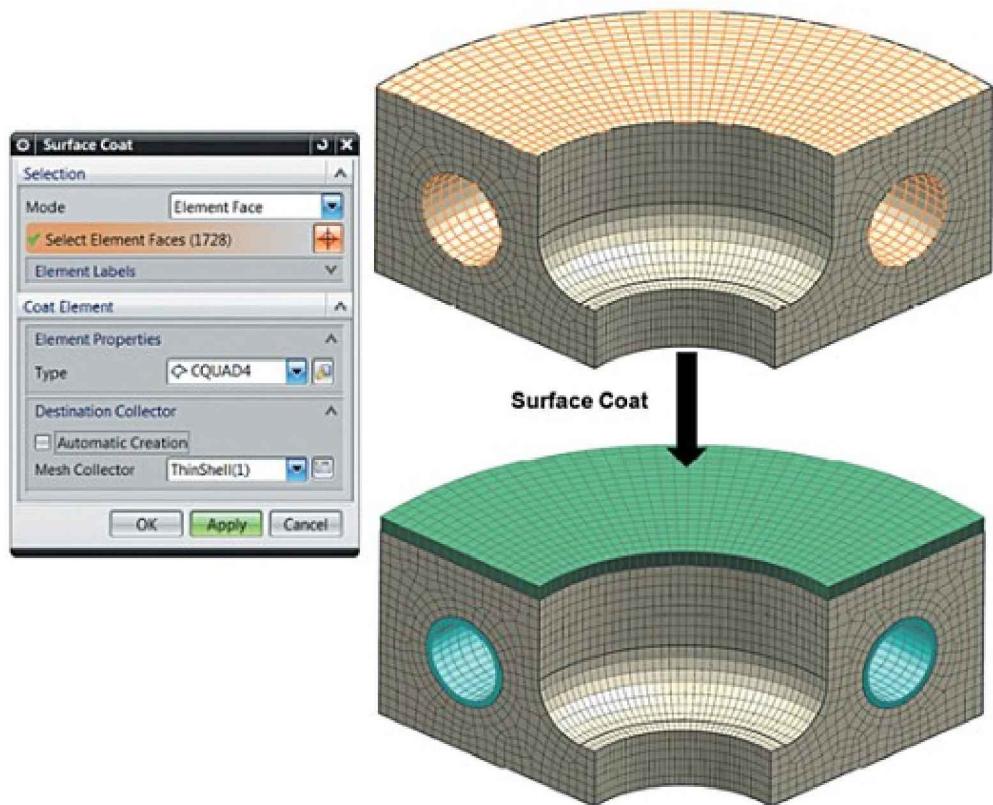


Figure 3.29. Creating a surface coat

3.3.3. Creating 1D elements

One-dimensional elements are often used to model beams, cables, and shock absorbers, or to account for stiffness and inertial properties that are not considered in the FE model as parts of the structure. For example, 1D

elements are frequently used to correctly transmit force and kinematic conditions from one part of the structure to another (Figure 3.30).

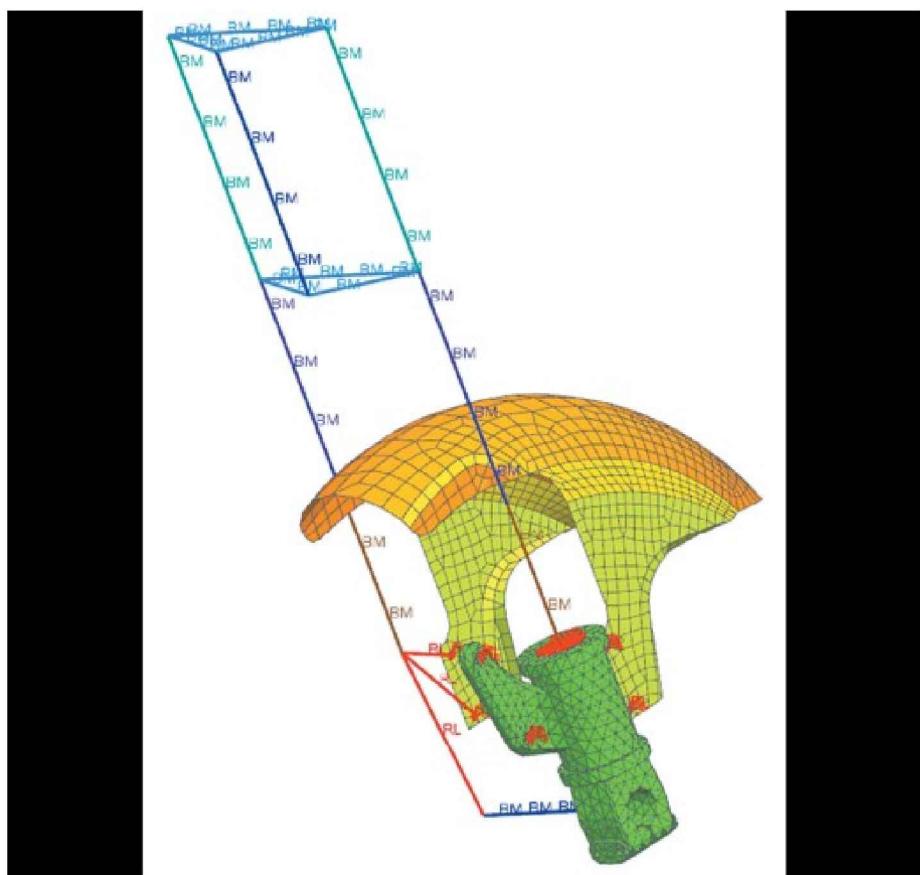


Figure 3.30. 3D-2D-1D finite-element model

Consider a list of the most important 1D elements for NX Nastran solver. There is no *Type of physical properties* for some element types in the following table, therefore properties of such elements must be specified in associated data of the meshes.

Element name	Description	Type of physical properties
CBAR	Simplified beam element with six degrees of freedom. Offset of the neutral axis relative to element nodes is allowed, as well as transverse shear effects.	PBAR PBARL
	Beam element with six degrees of freedom and cross sectional deformation. Centerline shift relative to element	PBEAM

CBEAM	nodes and transverse shear effects are allowed. The neutral line and the shear center do not have to coincide. Wedge shape of the element is allowed.	PBEAML
CBUSH	Elastic damper element	PBUSH
CDAMP1	Scalar damper element	PDAMP
CDAMP2	Scalar damper element	—
CELAS1	Scalar elastic element	PELAS
CELAS2	Scalar elastic element	—
CGAP	One-dimensional nonlinear contact element	PGAP
CMASS1	Scalar mass element	PMASS
CMASS2	Scalar mass element	—
CROD	One-dimensional tension-compression and torsion element. Two degrees of freedom per node	PROD
CONROD	One-dimensional tension-compression and torsion element. Two degrees of freedom per node	—
CTUBE	One-dimensional tube-like tension-compression and torsion element. Two degrees of freedom per node	PTUBE
CVISC	Viscous damping element	PVISC
PLOTEL	Dummy one-dimensional element for visualization	—
RBAR	Stiff rod element. Independent degrees of freedom can be defined at one or two nodes	—
RBE2	Stiff element with independent degrees of freedom defined for one node and dependent degrees of freedom defined for several nodes	—
RBE3	Interpolation element that defines the displacement of the master node as weighted mean of displacements of other nodes	—
RROD	Rigid hinged element with one degree of freedom	—

For a detailed description of physical properties and associated data of meshes for some element types, see [4, 8].

You can create 1D finite elements in NX Advanced Simulation using a broad selection of commands grouped in the following tools:

- **1D Mesh** creates one-dimensional finite elements based on edges and curves.

- ***1D Connection*** creates one-dimensional elements that link together geometry objects or finite-element objects.
- ***Element operations*** allow to manually create and manipulate elements that are not associated with geometry.

You can use the ***1D Mesh*** command to create geometry-associative meshes that consist of sets of 1D elements with two nodes per element that fully describe the position of elements in space. Depending on the element type, one of the most important properties is the orientation of the element along its axis. This orientation is determined using the ***1D Mesh*** command.

To create a mesh out of 1D elements based on geometry, open the ***1D Mesh*** dialog box and specify the following parameters (Figure 3.31):

- *Select Objects* to specify geometry curves or edges of polygonal faces to use in creating the 1D elements. When you select objects, pay attention to their direction because some elements, such as beam elements, have their own coordinate systems, the initial orientation of which depends on the specified direction of curves and edges. If you choose a wrong direction, you can change it at any time by editing the completed ***1D Mesh*** command or by reorienting the coordinate systems in associated data of the elements.
- When you select several linked curves/edges, you can maintain uniform direction by selecting the *Auto Chain Selection* check box.
- You can *Reverse Direction* to change the direction of selected objects to the opposite.
- Select *Type (Element Properties)* to specify the type of 1D element, for example beam *CBEAM*.
- Select *Mesh Density by, Number of Elements* to specify the number of elements on objects. You can enter either *Number of elements per object* or *element Size*.
- Select *Merge Nodes* to create a continuous mesh on several selected curves/edges using shared nodes at junction points with a specified tolerance.
- Use the *Destination Collector* group to select the collector that contains the physical properties of elements. Alternatively you can specify an existing collector from the list (clear the *Automatic*

Creation check box) or specify automatic collector creation, or alternatively create a new collector right in the current dialog box.

- You can preview the arrangement of nodes on selected curves or edges using the *Boundary Nodes* option.

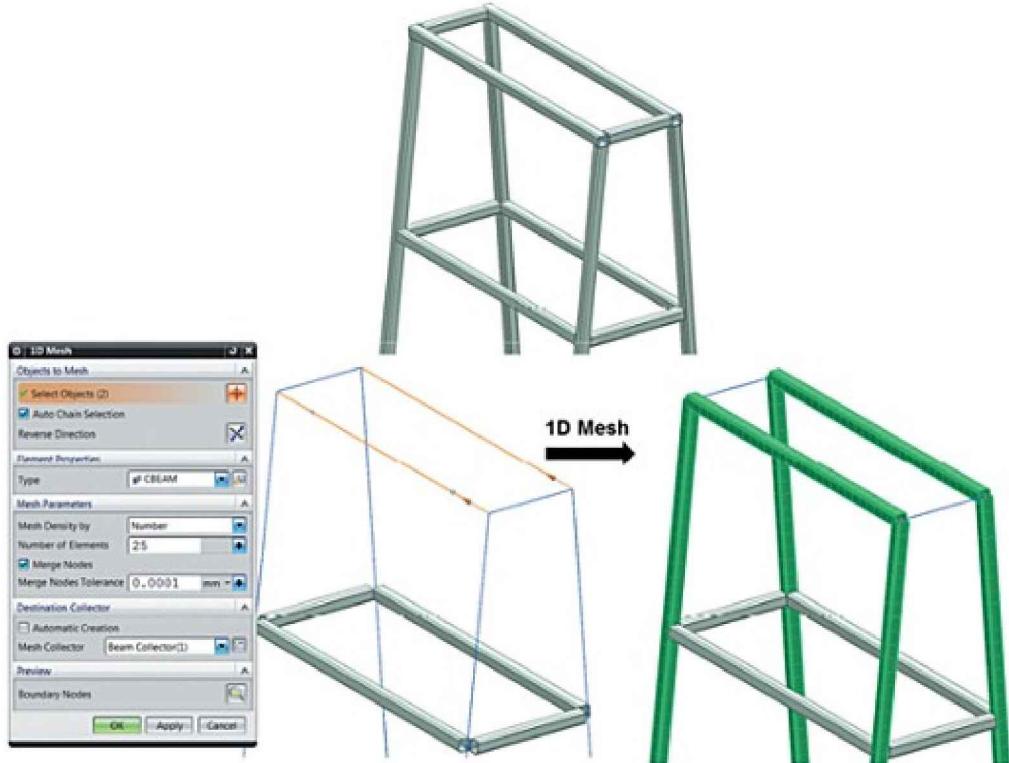


Figure 3.31. Creating a 1D mesh

You can create individual one-dimensional elements or sets of such elements using tools (Figure 3.32) that allow to use a relevant command to quickly and efficiently create the required 1D finite-element submodel that connects elements with other elements and correctly transmits kinematic and force factors. An indisputable advantage is the consideration given to association of finite-element objects with geometry objects. This allows automatically updating the created 1D connections whenever geometry is modified.

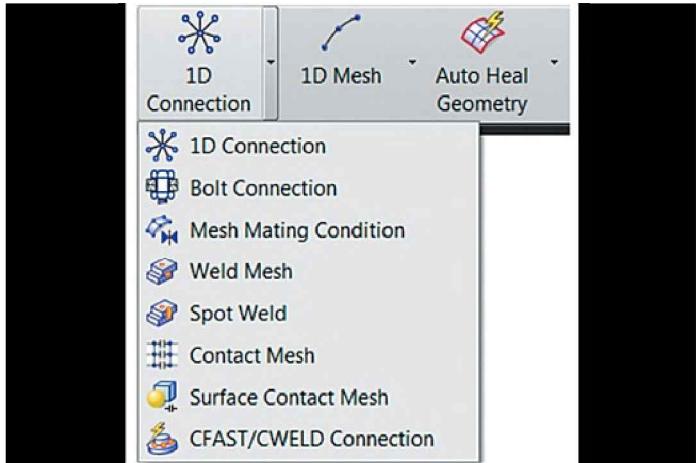


Figure 3.32. 1D connection creation tools

You can create one-dimensional finite elements through nodes/elements or geometry objects using the **1D Connection** command in one of the following ways:

- Select *Node to Node* to create one-dimensional elements through the selected nodes along the shortest distance.
- Select *Element Edge to Element Face* to create RBE2 and RBE3 elements connecting nodes of the selected element edges to nodes of the specified element faces. This is equivalent to specifying constraint equations for degrees of freedom.
- Select *Point to Point* to create 1D elements through selected points. Nodes created at these points are used in building the mesh (Figure 3.33, A).
- Select *Point to Edge* to create 1D elements through the specified point or node and all nodes lying on the specified edge (Figure 3.33, B).
- Select *Point to Face* to create 1D elements through the specified point or node and all nodes lying on the specified face (Figure 3.33, C).
- Select *Edge to Edge* to create 1D elements through nodes of an edge and nodes of another selected edge along the shortest distance (Figure 3.33, D).
- Select *Edge to Face* to create RBE2 and RBE3 elements connecting nodes of the selected edges to nodes of the specified faces. This is equivalent to specifying constraint equations for

degrees of freedom.

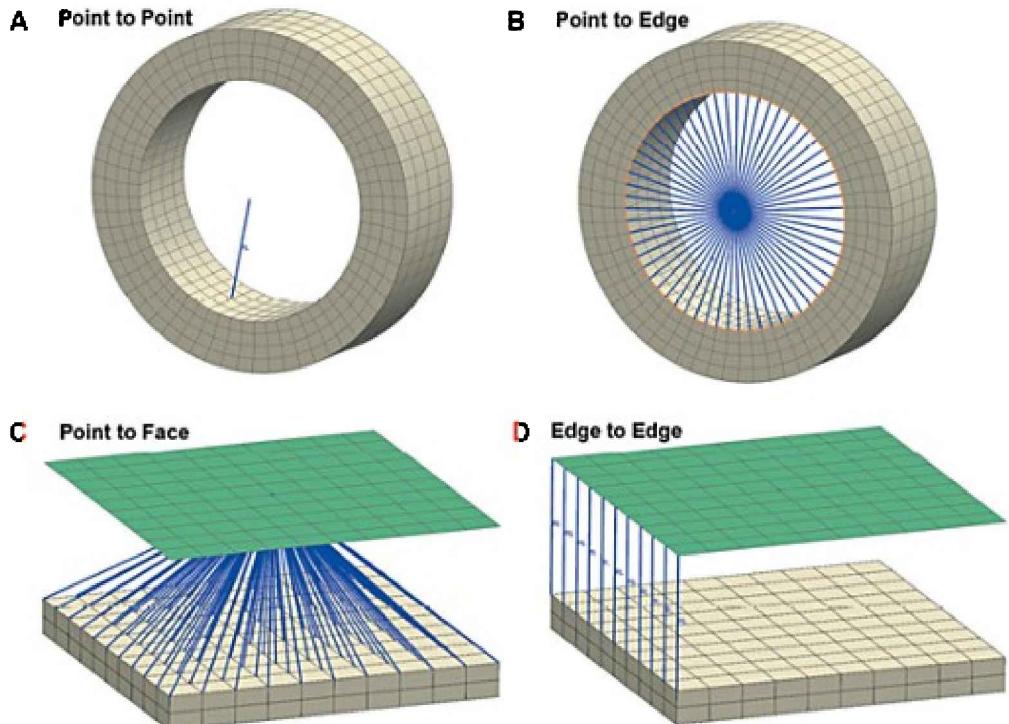


Figure 3.33. Methods of creating 1D connections:

A. Point to Point. B. Point to Edge. C. Point to Face. D. Edge to Edge

If constraints were initially based on geometry objects, then in addition to creating a 1D collector with new 1D elements in the model tree, the so-called *Connection Collectors* are created, containing information about creation parameters of relevant connections (Figure 3.34). These parameters can be edited.

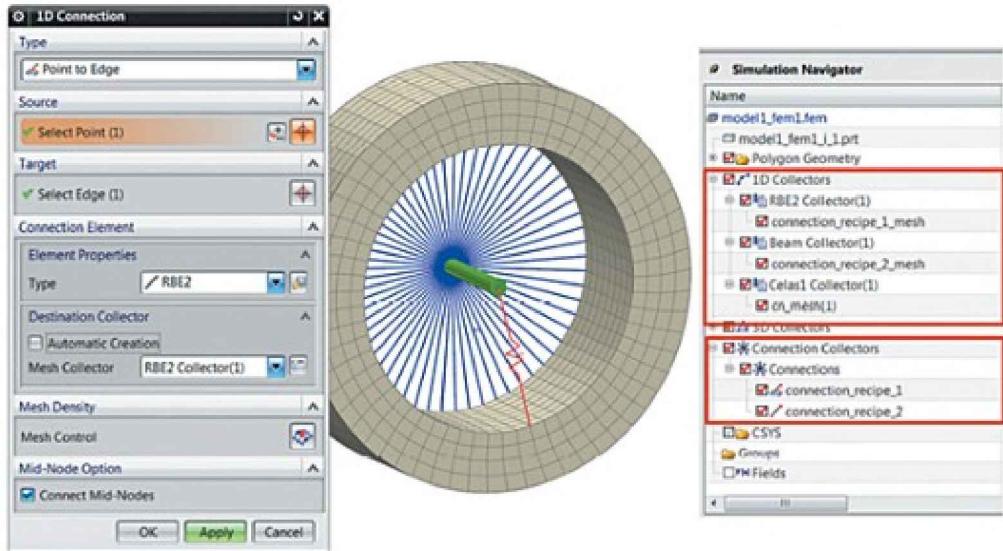


Figure 3.34. 1D Connection command

To create connections from 1D elements, open the **1D Connection** dialog box and specify the most important parameters (Figure 3.34):

- Select a *Type* to specify the method used to create connections.
- Select *Source* and *Target* to specify objects that will serve as the basis for 1D connections. The object types depend on the connection type you specify.
- Use options in the *Connection Element* group to specify the *Type* of 1D elements and the *Destination Collector*, that is, physical properties of the elements. The *Destination Collector* corresponds to the collector containing the physical properties of elements.
- Use the *Mesh Control* option to specify the number of 1D elements that you want to create between two nodes.
- Select *Mid-Node Option* to include mid-nodes of connecting 2D or 3D finite elements when you create 1D connections based on edges and faces.

If you need to specify connections using one-dimensional elements, you should do this based on geometry models. In this way, whenever the mesh is modified, connection information is not lost, and 1D elements are automatically recreated (Figure 3.35).

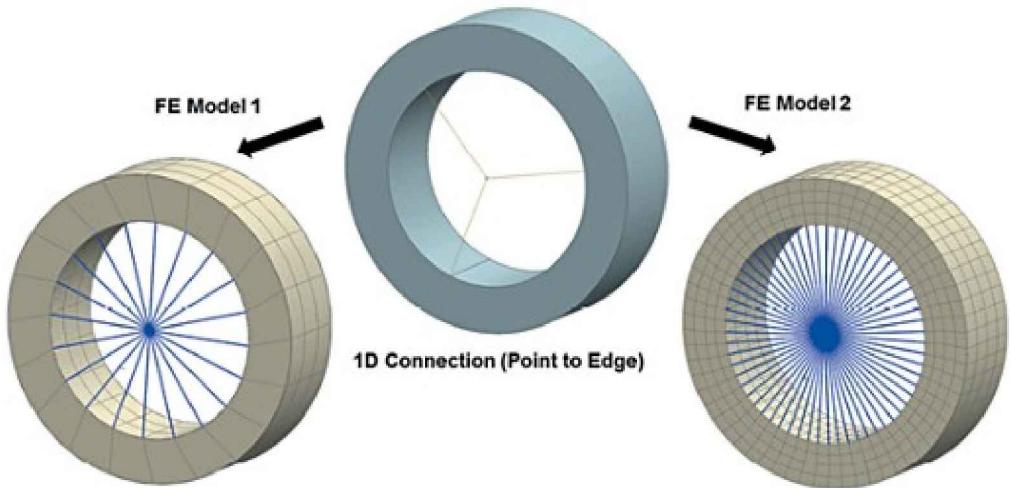


Figure 3.35. Connections based on geometry objects

To correctly model bolted connections between two parts of a structure, you can create a set of one-dimensional elements. The body of the bolt is typically modeled as a beam element, and the interaction of the head or nut with the structure is modeled by connection elements (RBE2, RBE3), which provide a correct force arrangement for the region of the bolted connection (Figure 3.36). In some cases you also need to account for the contact interaction of the joined parts of the structure. To create an arrangement of this kind, you can repeatedly apply standard 1D element creation commands or use the specialized **Bolt Connection** command. You can use this command to quickly create several bolted connections at once. These are specified based on geometry objects, so the connections you create are preserved in case the meshes are modified.

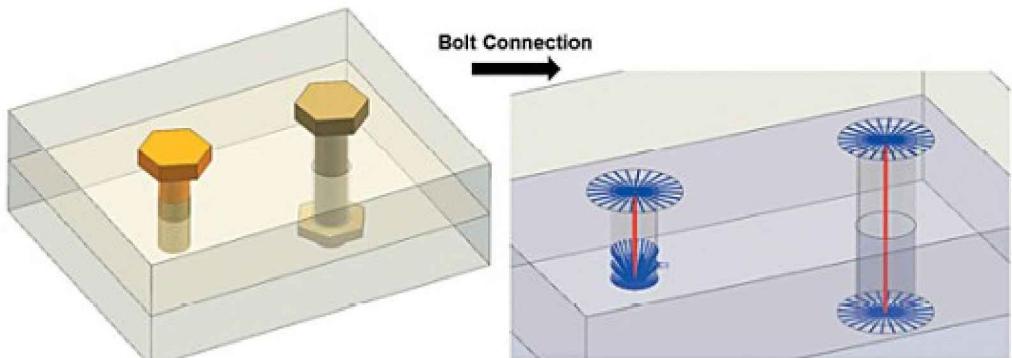


Figure 3.36. Modeling bolted connections

There are the following connection types:

- *Bolt with Nut*, Figure 3.37, A. Spider-type connections (sets of 1D elements radiating from a single point) are created on surfaces where the head and the nut contact the structure, while the shank of the bolt is modeled by the selected 1D element, usually a beam element.
- *Bolt in Tapped Hole*, Figure 3.37, B. Spider-type connections are created on surfaces where the head contacts the structure, and on the surface of the hole along the thread, while the shank of the bolt is modeled by the selected 1D element.
- *Spider at Junction*, Figure 3.37, C. Two spider-type connections are created, all 1D elements radiating from a single specified point.

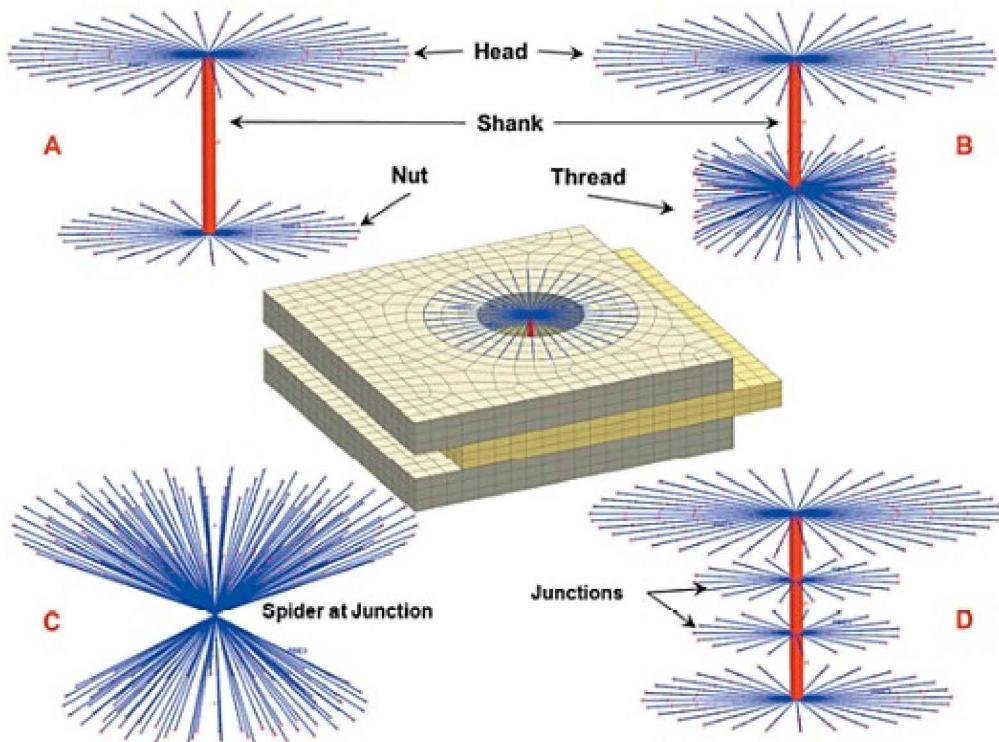


Figure 3.37. Types of bolted connections

When you model a bolt with a nut or thread, and need to take into account the intermediate part of the structure and the way it engages the bolted connection, you can create a spider-type connection at the specified point between the head and the nut or between the head and the thread (Figure 3.37, D).

So, to create bolted connections from 1D elements, you need to open the **Bolt Connection** dialog box. The layout of the dialog box depends on the selected type of connections. A summary of the most important parameters is provided in the following table.

Type	
Type	Selection of bolted connection type <i>Bolt with Nut</i> <i>Bolt in Tapped Hole</i> <i>Spider at Junction</i>
Head/Nut	
<i>Define Head By/</i> <i>Define Nut By/</i> <i>Define Junction1 By/</i> <i>Define Junction2 By)</i>	Specify the type of geometry objects used to define the head, the nut, or intermediate junctions. For a bolt in tapped hole, only the head is specified here. Specify <i>Hole Edge</i> and select edges of the bolt hole. Specify <i>Center Point</i> and select the center point of the edge of the bolt hole and the corresponding face.
Tap	
<i>Spider Diameter</i> <i>Spider Diameter</i>	Select the method used to specify the diameter of the spider elements. Select <i>Percent of Hole</i> to specify the diameter as a percentage of the bolt hole, a 100% value means the spider diameter is the same as the hole diameter. Select <i>Diameter</i> to specify the spider diameter.
<i>Select Tapped Surface</i> <i>Bolt Length</i> <i>Effective Thread Length</i>	Specify the cylindrical surface of the bolt hole. Specify the length of the entire bolt. Specify the tapped length of the bolt.
Use Spring element	
<i>Use Spring element to connect Head to Nut/Use Spring element to connect Head to Tap</i>	Modeling the bolt shank by a zero-length elastic element instead of a beam element.

Junction Plane	
Create Spider at Junction Plane	Controls the creation of an extra spider-type junction on an intermediate plane between the head and the nut or between the head and the thread.
Shank Element/Spider Connection	
Type	Specify a 1D element type used for modeling the bolt shank (typically a beam element) and the type of spider elements (typically RBE2 and RBE3). Keep in mind the associated data of elements
Destination Collector	You can use the <i>Destination Collector</i> option to specify the collector that contains physical data of the elements, or select an existing collector from the list (clear the <i>Automatic Creation</i> check box), or specify automatic creation of a collector (select the <i>Automatic Creation</i> check box), or create a new collector right in the dialog box.
Connect Spider to Midnodes	Controls whether intermediate nodes of mesh elements of mated parts of the structure are respected when creating 1D elements of the spider.

You can use the **Spot Weld** command to create one-dimensional elements that create a pairwise connection of one set of points to another, for example to create rivet joints. It involves creating so-called mesh points where the nodes are created during mesh generation (Figure 3.38). When you use this command, specify either sets of points that define the nodes of 1D elements when projected on selected faces, or edges/curves with the specified mesh density parameter to define the number of nodes along the selected objects. Also specify the type of 1D elements and the corresponding physical properties.

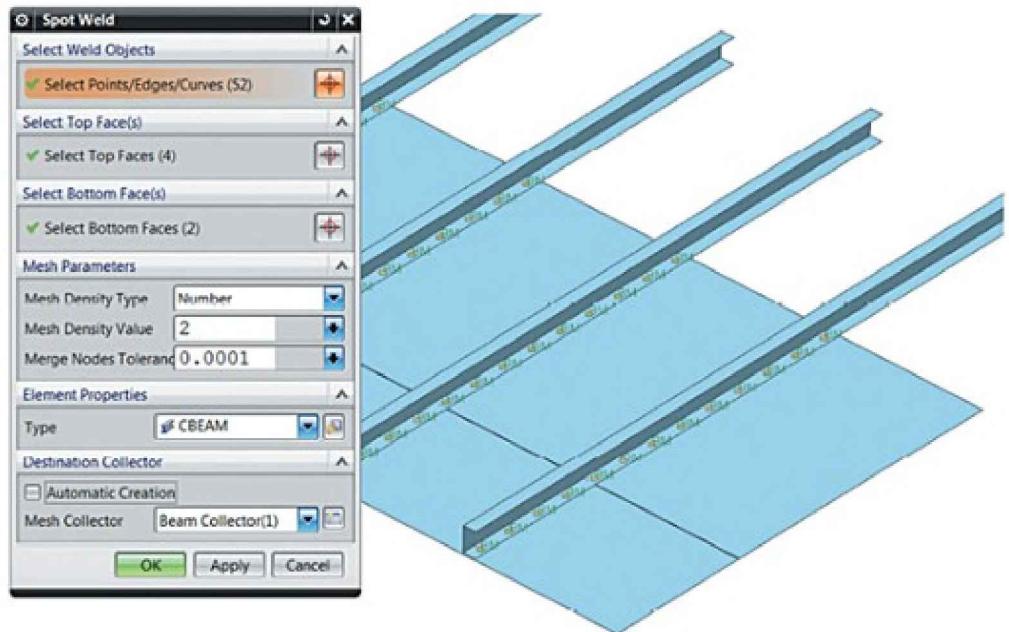


Figure 3.38. Creating a spot weld

Use the **CFAST/CWELD Connection** command as an alternative to **Spot Weld** to model spot welding with CFAST and CWELD elements that are created based on the specified point sets and nodes with the specified physical properties (Figure 3.39).

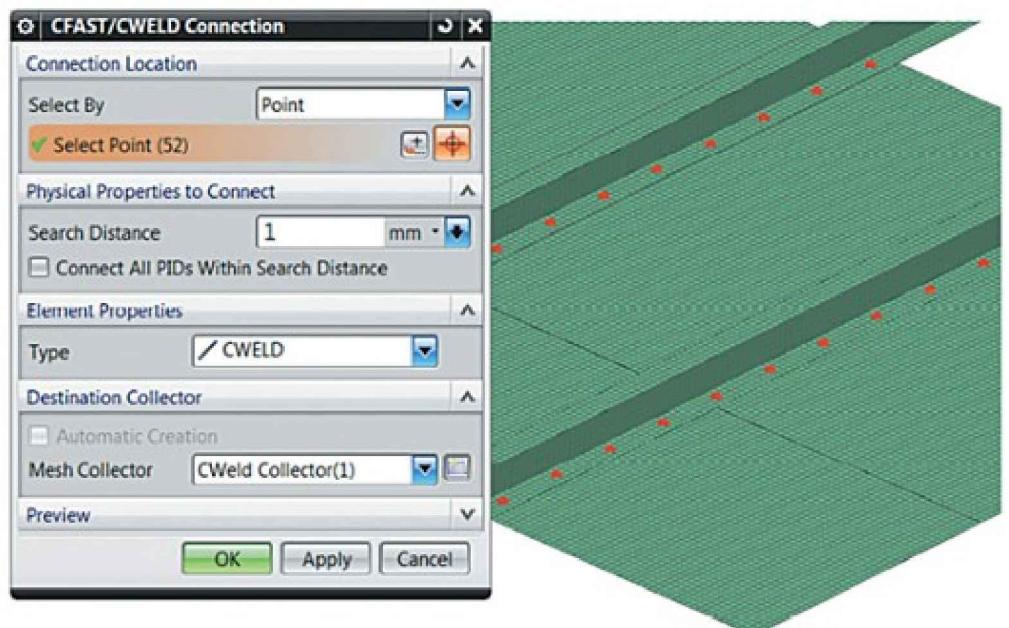


Figure 3.39. Creating a spot weld

You can use the **Contact Mesh** and **Surface Contact** commands to facilitate creation of gap-type contact elements (CGAP) between geometry objects (edges and surfaces respectively). These commands are used only with geometry-associated meshes. You can use them to create pairs of GAP elements through nodes of selected edges for axisymmetric calculations and through nodes of selected faces for spatial calculations (Figure 3.40).

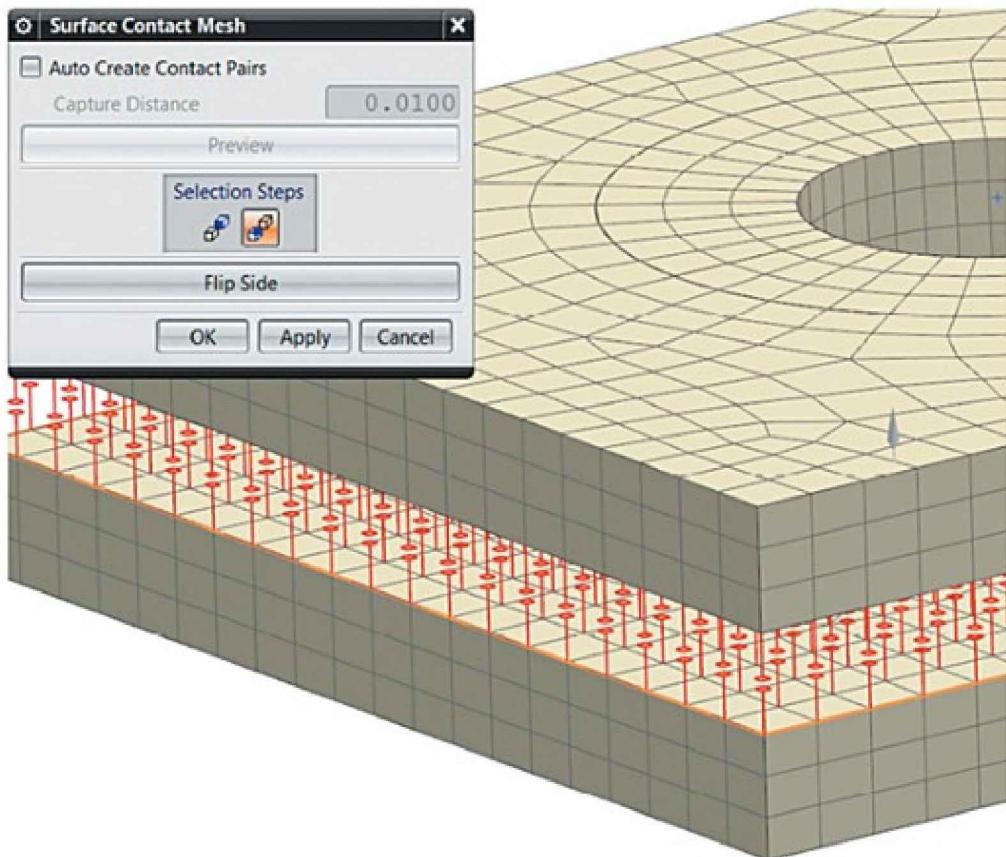


Figure 3.40. Creating a contact mesh

3.3.4. Creating 0D elements

0D elements are, in essence, scalar elements that have only one node. The most common use case for these elements is adding concentrated masses to FE models to account for a neglected inertial component. 0D elements are specified at points or nodes that can be connected to the FE model via 1D elements or be integral to the model.

Consider a list of the most important 0D elements for NX Nastran solver. If properties of some elements are not included in the following physical property table, these elements must be specified through the associated data of meshes.

Element name	Description	Type of physical properties
CBUSH	Elastic damper element	PBUSH
CDAMP1	Scalar damper element	PDAMP
CDAMP2	Scalar damper element	—
CELAS1	Scalar elastic element	PELAS
CELAS2	Scalar elastic element	—
CMASS1	Scalar mass element	PMASS
CMASS2	Scalar mass element	—
CONM1	Symmetric (6x6) mass matrix at the node	—
CONM2	Concentrated mass at the node	—
CWELD	Spot weld element, physical properties determined in accordance with the specified material	PWELD
CFAST	Spot weld element, physical properties specified directly	PFAST

You can create 0D finite elements using the following commands:

- Use **0D Mesh** to create 0D finite elements based on mesh points, created nodes of the mesh and geometry objects: points, edges, curves, faces, and bodies.
- Use **Create Element** and **Element operations** to create 0D manually based on nodes.

To create a mesh out of 0D elements based on geometry, open the **0D Mesh** dialog box and specify the following parameters (Figure 3.41):

- Use **Select Objects** to specify mesh points, mesh point sets, points, edges, curves, faces, and bodies.
- Select **Type (Element Properties)** to specify the type of 0D element, for example concentrated mass.
- Select **Mesh Density, Number of Elements** to specify the number of elements per object. You can enter either **Number of elements per object** or **element Size**. This parameter is meaningful only if the selected objects are not connected to other elements, otherwise all 0D elements are created at existing nodes.

- Use options of the *Destination Collector* group to set up options similar to options of other simulation mesh creation commands.
- You can preview the arrangement of nodes on selected objects using the *Boundary Nodes* option.

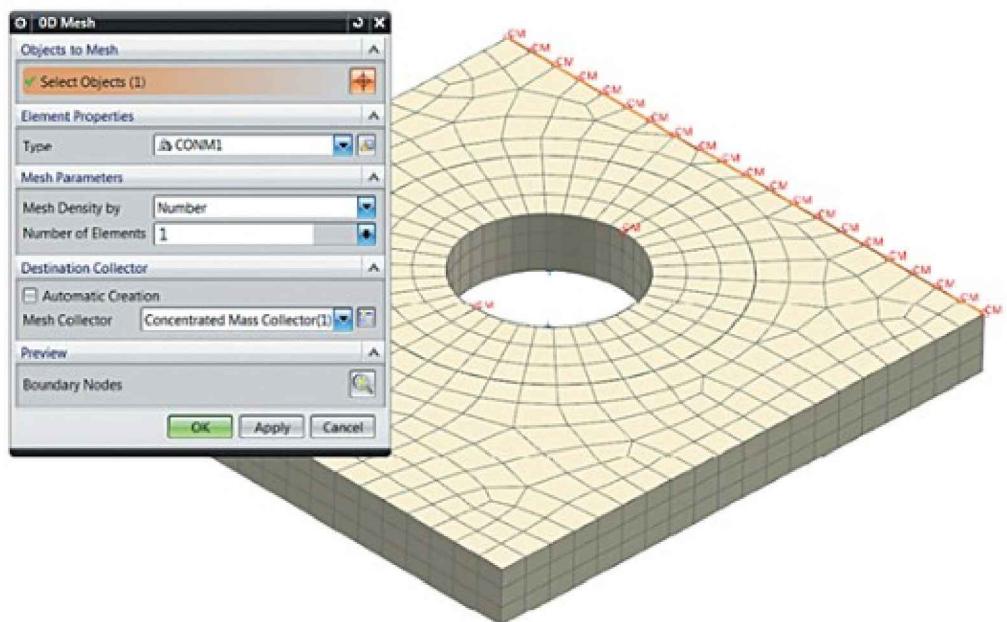


Figure 3.41. Creating 0D elements

3.3.5. Operations on nodes and elements

There are additional tools you can use to work with the finite-element mesh. These tools allow creating new nodes and elements directly, modifying existing nodes and elements, deleting nodes and elements, splitting elements, editing physical properties of individual elements, viewing relevant information about elements and much more. Note that when you use a command that modifies the FE mesh based on geometry objects, the mesh becomes locked for update. If the mesh is unlocked and updated, it is rebuilt, and all parameters set by the command that was used to create the mesh based on geometry are applied. All operations on nodes and elements that were performed based on mesh elements, are canceled.

Figure 3.42 shows the FE model node operations toolbar.



Figure 3.42. Commands of the Node Operations toolbar

The commands belong to three groups: creation, modification, and management.

You can create nodes using the following commands, all of which allow to control the numbering of new nodes:

- Use the ***Node Create*** command to create a node in space relative to the specified coordinate system. Referencing existing objects is recommended.
- Use the ***Node Between Nodes*** command to create a certain number of nodes between two selected nodes. Creation of a single node centered relative to three selected nodes is shown in Figure 3.43, A.
- Use the ***Node on Point/Curve/Edge*** command to create several nodes on a selected geometry object. You can specify the quantity of new nodes or the distance between nodes (Figure 3.43, B).

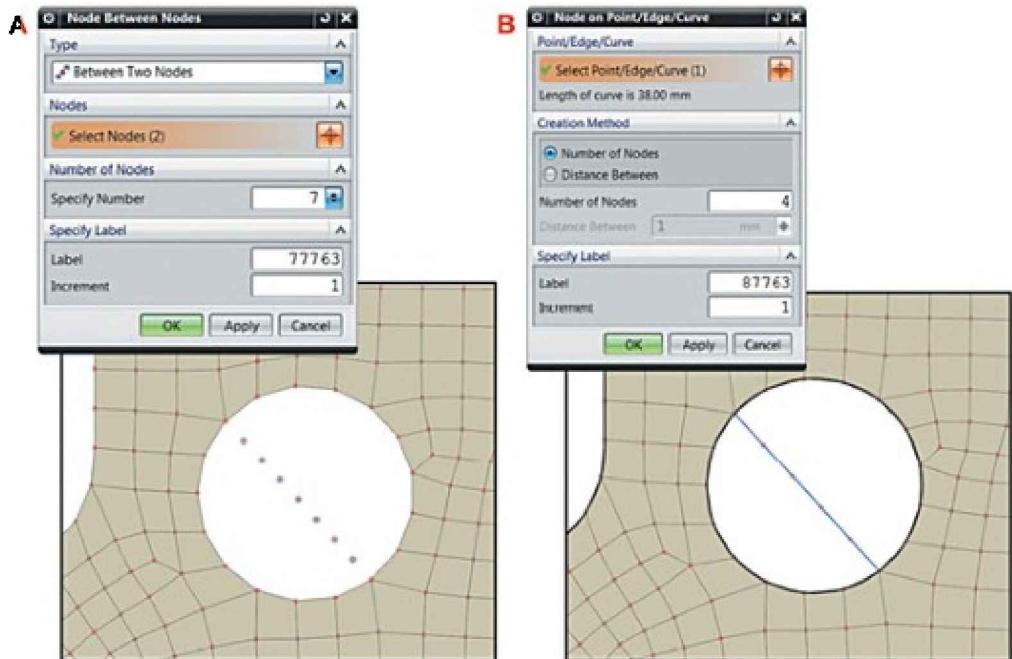


Figure 3.43. Creating nodes

You can change node location or copy nodes using the following commands. If you move any nodes, the mesh is automatically modified as well.

- Use the ***Node Translate*** command to copy or move a set of nodes: for a specified increment in the selected coordinate system and for a directional increment. You can also use space distribution following a formula or a table. This method is typically used to create a new finite-element model based on a deformed state produced by static or dynamic analysis (Figure 3.44, A).
- Use the ***Node Rotate*** command to copy or move a set of nodes by a specified angle relative to a selected axis.
- Use the ***Node Reflect*** to copy or move a set of nodes by mirroring it against a specified plane.
- Use the ***Node Drag*** command to interactively move a node by dragging it or by referencing it to geometry or space.
- Use the ***Node Align*** command to move nodes to an imaginary line defined by two other nodes (Figure 3.44, B).
- Use the ***Node Modify Coordinates*** command to move a node by specifying new coordinates in the selected coordinate system.

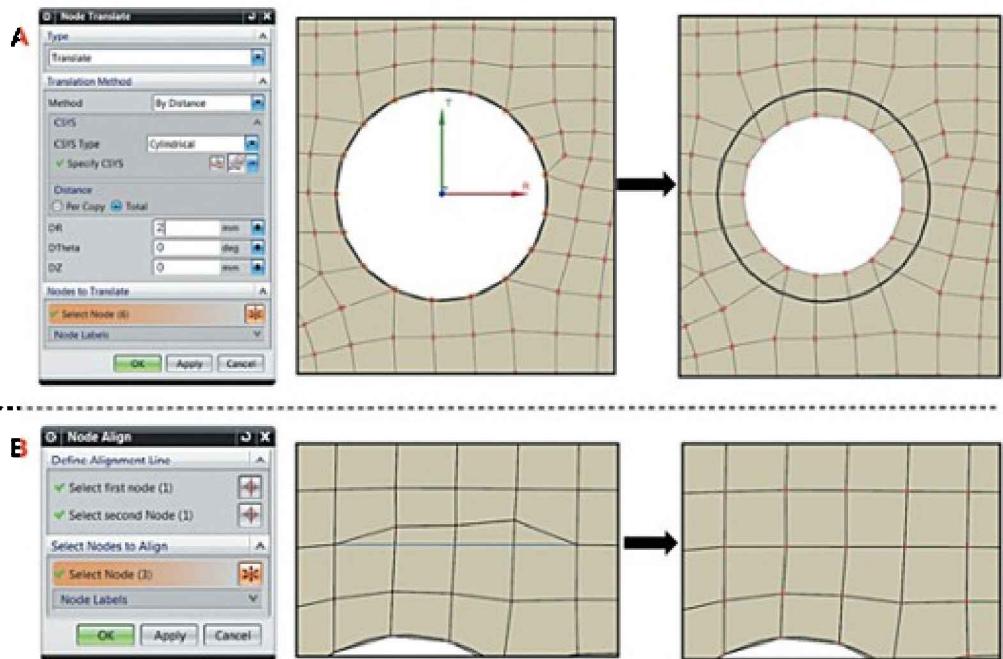


Figure 3.44. Moving nodes

The following commands allow managing nodes and viewing node information

- Use the **Assign Nodal Coordinate System** command to assign a new nodal coordinate system to nodes. You can select nodes by geometry objects they belong to. The nodal coordinate system is very important for specifying loads and constraints.
- Use the **Node Modify Label** to renumber selected nodes by shifting their numbers or assigning a new unused number to a node.
- Use the **Node Delete** command to delete selected nodes that do not belong to elements.
- Use the **Node/Element Information** command to view textual information concerning selected nodes: node numbers, coordinate systems, coordinates, and element numbers.
- Use the **Nodal Coordinate System** command to identify the coordinate system of selected nodes.

Figure 3.45 shows the FE model element operations toolbar.



Figure 3.45. Commands of the Element Operations toolbar

All element operation commands belong to three groups, just like node commands: creation, modification, and management.

You can create elements with the following commands, which allow to assign numbers to new elements:

- Use the **Element Create** command to create 0D, 1D, 2D, or 3D elements node-by-node with the selected element type and destination collector (physical properties).
- Use the **Element Extrude** command to create a set of elements by extruding edges (creating 2D elements) or faces (creating 3D elements) of existing elements. You can create several elements at once by specifying the necessary number. You can extrude by specifying direction, a path, or by projecting onto a surface (Figure 3.46). The operation is performed for the selected element type and destination collector (physical properties).
- Use the **Element Revolve** command to create a set of elements by revolving edges (creating 2D elements) or faces (creating 3D elements) of existing elements around a selected axis with a specified angle. You can create several elements at once by specifying the necessary number. The operation is performed for the selected element type and destination collector (physical properties).
- Use the **Element Copy and Translate** command to copy the whole element, an edge of the element, or a face of the element in order to create a copy of the source element, a 1D element, or a 2D element. You can copy along a direction or by specifying an increment in the selected coordinate system. You can create several elements at once by specifying the necessary number. The operation is performed for the selected element type.
- Use the **Element Copy and Project** command to create 1D and 2D elements by projecting 1D elements, 2D elements, element

edges, or element faces onto the selected face. The operation is performed for the selected element type.

- Use the **Element Copy and Reflect** command to copy the whole element, an edge of the element, or a face of the element in order to create a copy of the source element, a 1D element, or a 2D element. The element is copied by reflecting it in a plane.
- Use the **3D Sweep Between** command to create 3D hexahedral elements between two surfaces with a specified number of element layers. The operation is performed for the selected element type and destination collector (physical properties).

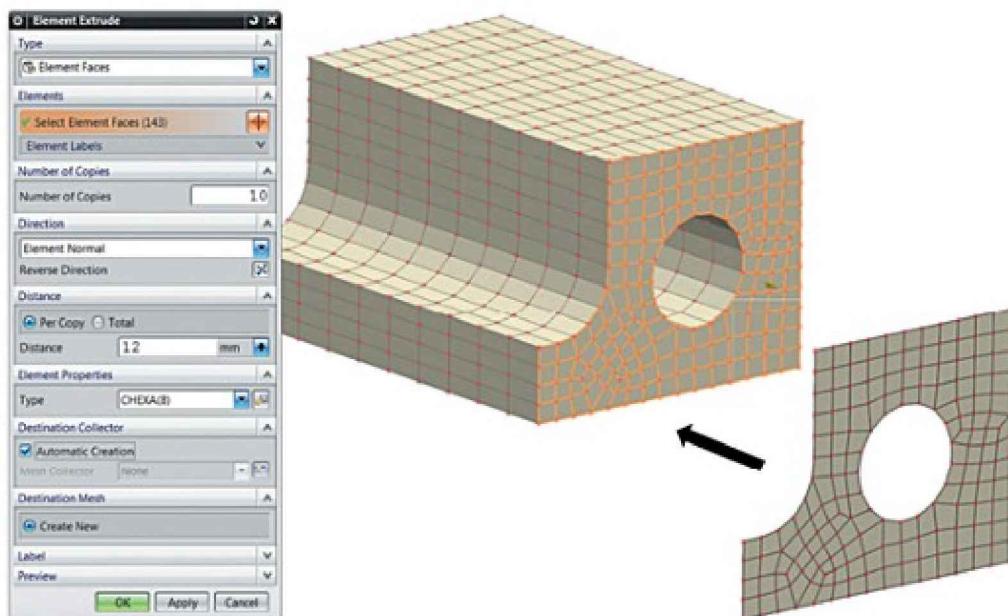


Figure 3.46. Creating elements

Elements can be modified using the following commands:

- Use the **Element Modify Order** command to change the order of selected elements from linear to quadratic and vice versa. You can also change the method of creating midnodes (linear, curvilinear or mixed).
- Use the **Split Shell** command to split 2D quadrangular and triangular elements into several triangular elements using one of eight methods (Figure 3.47, A). This command is used to replace quadrangular elements that do not fulfill quality criteria with

triangular elements.

- Use the **Combine Triangles** command to combine two selected adjacent triangular elements into one quadrangular element (Figure 3.47, B).
- Use the **Move Node** command to specify coincidence of the selected node of an element with a node of another element.
- Use the **Element Modify Connectivity** command to rebuild an element by replacing one of its nodes with another node that is not part of any neighbor elements.

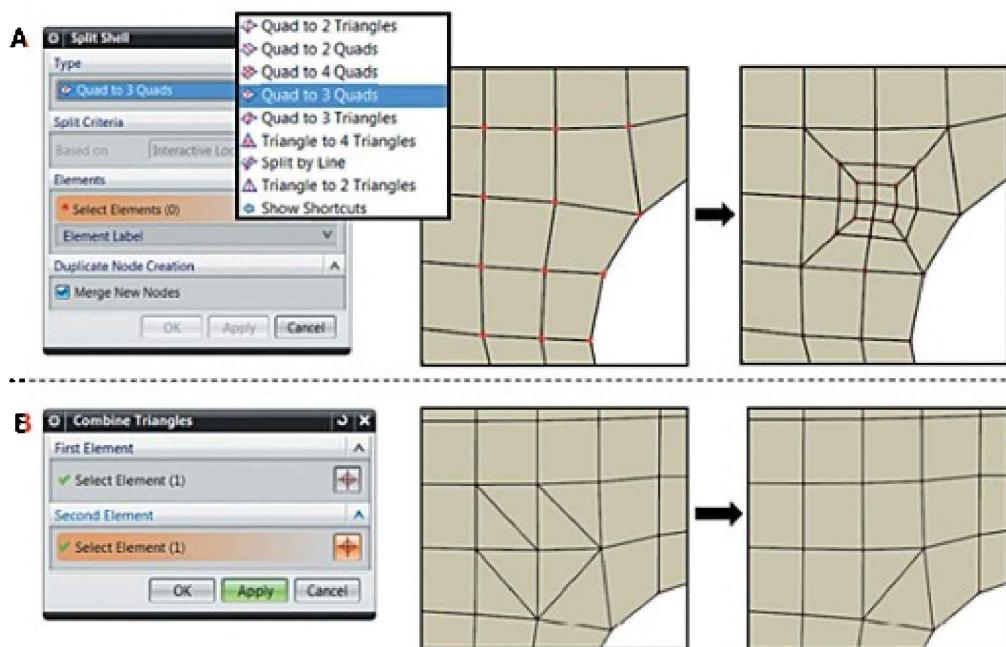


Figure 3.47. Editing elements

You can use the following commands to manage elements and element properties

- Use the **Element Modify Label** command to renumber selected elements by specifying a number offset or by assigning a new unused number to an element.
- Use the **Edit Mesh Associated Data** command to assign new associated data to individual elements. For example, for shell elements you can modify thickness and material orientation, for beam elements you can specify cross section orientation, and for elastic elements you can modify stiffness values. This command is

useful when you need to modify properties of individual mesh elements without splitting the mesh into element sets.

- Use the **Element Delete** command to delete elements.
- Use the **Element Extract** to extract a set of elements from an existing mesh to a new mesh. You can use this command if you need to modify the physical properties of elements in the new mesh (Figure 3.48).
- Use the **Node/Element Information** command to view textual information concerning selected elements: element numbers, element types, names of meshes, names of collectors and nodes that belong to these collectors.
- Use the **Unlock Mesh** to unlock the mesh for subsequent updating using parameters that were created when the mesh was built based on geometry. Locking is applied to all meshes with elements that had nodes and elements operated upon.

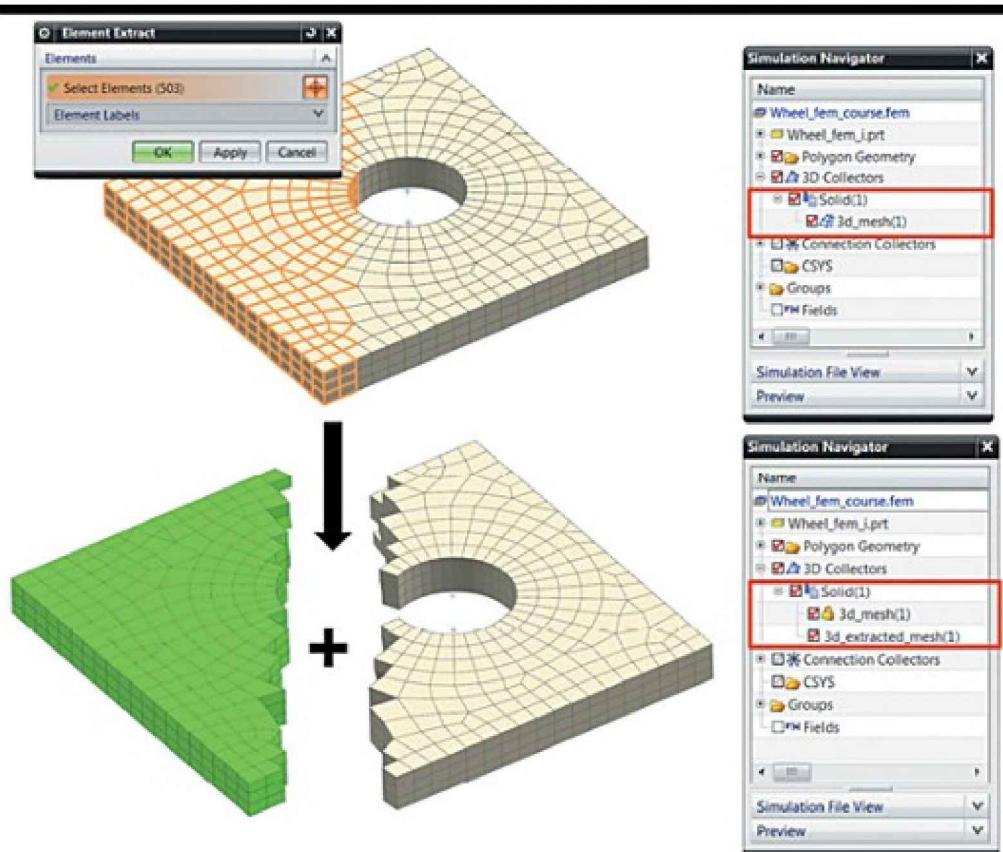


Figure 3.48. Extracting elements

3.3.6. Mesh mating conditions

An important FE model creation tools is ***Mesh Mating Condition***, which ensures correct interaction between elements of different meshes. Depending on the situation, you can specify conditions for: creating a continuous mesh at the body split boundary, creating a discontinuous mesh at the split boundary of bodies with identical node locations, and creating a discontinuous mesh between bodies with displacement constraints.

Mesh mating conditions are used only for geometry objects. These conditions allow to connect:

- A body with another body
- A sheet body (midsurface) with another sheet body
- A sheet body with another body

A typical use case of mesh mating conditions is shown in Figure 3.49. It shows a structure containing a single complex body. To design an FE model, the structure must be split into several bodies (Figure 3.49, A). To create an FE model of the structure, meshes for several bodies are created, however finite elements in body split areas are not connected (Figure 3.49, B), so the results are incorrect. Body mating conditions are defined in areas where the bodies are joined together (Figure 3.49, C). The FE model needs updating because these conditions impose new constraints on the created mesh. When the meshes are rebuilt, common nodes and corresponding common faces of elements appear on touching surfaces of separate bodies. This means that the FE model is continuous (Figure 3.49, D).

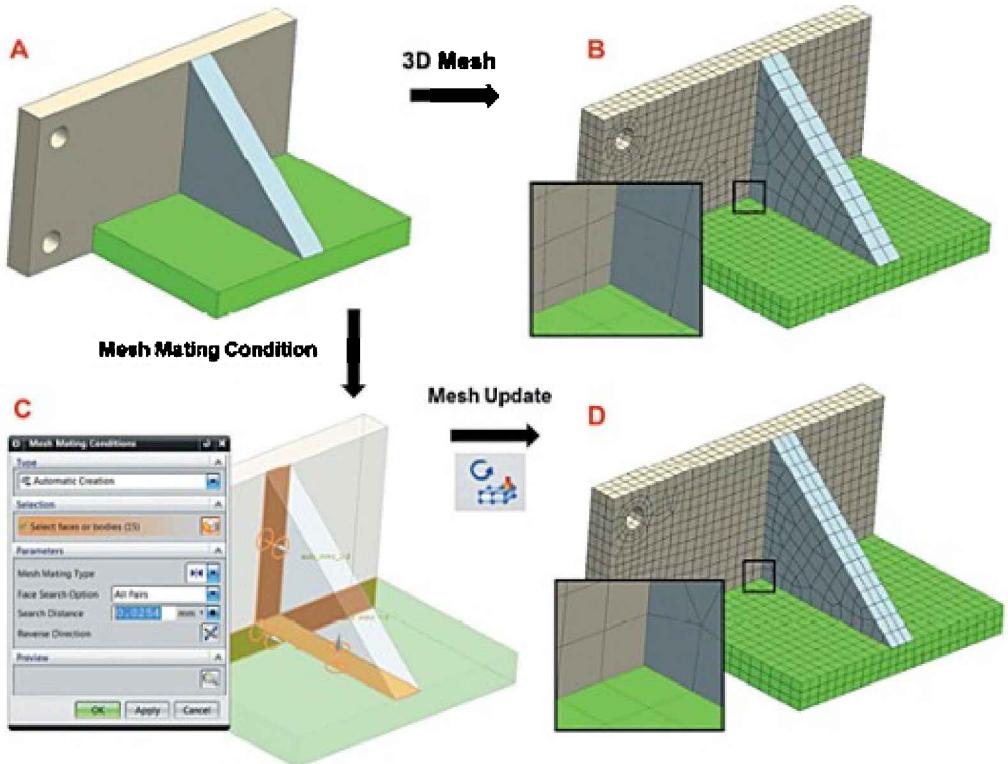


Figure 3.49. Example of mesh mating conditions

Mating conditions are imposed on pairs of faces. There are three possible mesh mating types for the selected faces (Figure 3.50):

- Using *Glue Coincident* results in an identical mesh on faces with shared nodes. This is achieved by projecting one face onto the other and splitting the faces to make touching faces identical.
- Using *Glue Non-Coincident* results in overlap of coupling conditions (MPC) of the two faces' nodes.
- Using *Free Coincident* results in an identical mesh on faces that have distinct nodes with the same location (duplicate nodes). One face is projected onto the other, and the faces are split to make the touching faces identical. This coupling type is used if you need to specify contact interaction conditions afterwards.

You can create mesh mating conditions in the ***Mesh Mating Condition*** dialog box by specifying the following parameters (Figure 3.50):

- In the **Type** list, select the method used to specify conditions:
 - *Automatic Creation*: find touching objects among the given set

automatically.

- *Manual Creation*: specify the objects manually.
- In the *Selection* group, select objects that need to have meshes mated (*Manual Creation* type), or specify a set of objects (*Automatic Creation* type) among which the touching of faces will be determined with a given separation tolerance.
- In the *Mesh Mating Type* list, select one of the three mesh mating type described above. Note that you can change the mating type after creating the conditions and updating the meshes.
- In the *Face Search Option* list and the *Search Distance* box, set up options for finding touching faces automatically.

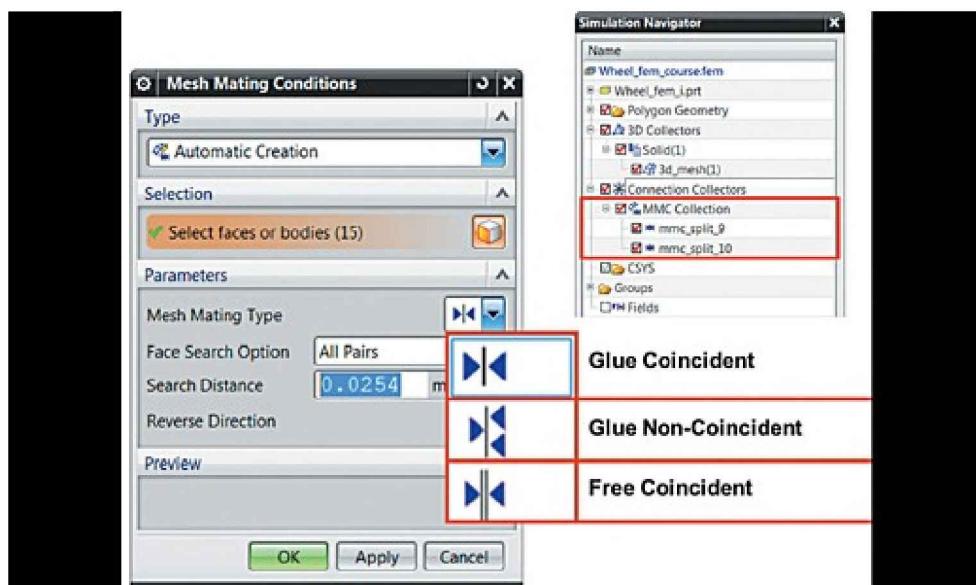


Figure 3.50. Creating mesh mating conditions

3.4. Associated data of finite elements

If you use elements to design a finite-element model, remember that you need to specify their physical properties in the corresponding mesh collectors. For some elements, there are so-called Element Associated Data that contain additional properties of individual elements and Mesh Associated Data that also contain additional properties, but these properties apply to a set of elements that form a mesh.

You can modify associated data of individual elements using the **Edit Element Associated Data** command on the **Element Operations** toolbar.

The usual way to modify associated data of meshes is using the ***Mesh Associated Data*** command on the **Advanced Simulation** toolbar or the ***Edit Mesh Associated Data*** command. To use the latter command, right-click the mesh in the model tree on the **Simulation Navigator** tab of the Resource Bar. This paragraph provides a detailed discussion of the data modification command as applied for different meshes. Here is the list of NX Nastran element properties that have associated data.

Element type	Associated data
CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR	Material Orientation Shell Thickness Shell Offset
CQUADX4, CQUADX8, CTRAX3, CTRAX6, CTRIAX6	Material Orientation
CTETRA, CHEXA	Material Orientation
CONM1, CONM2	Mass
CBAR , CBEAM	Section Orientation Section Offset Degrees of freedom of element ends
CELAS2	Spring Stiffness Damping Coefficient Stress Coefficient
CBUSH	Element Orientation Offset

CGAP	Element Orientation
CONROD	Material Area Torsional Constant Torsional Stress Coefficient Non-Structural Mass

3.4.1. Spatial and shell elements

If you need to respect anisotropic properties of a material that has different physical and mechanical properties in different directions, material orientation of 3D and 2D elements needs to be controlled. To achieve this, use the ***Edit Mesh Associated Data*** command and specify the new

material orientation for elements using one of the reorienting methods (Figure 3.51). Note that the initial material orientation in elements coincides with the coordinate systems of these elements.

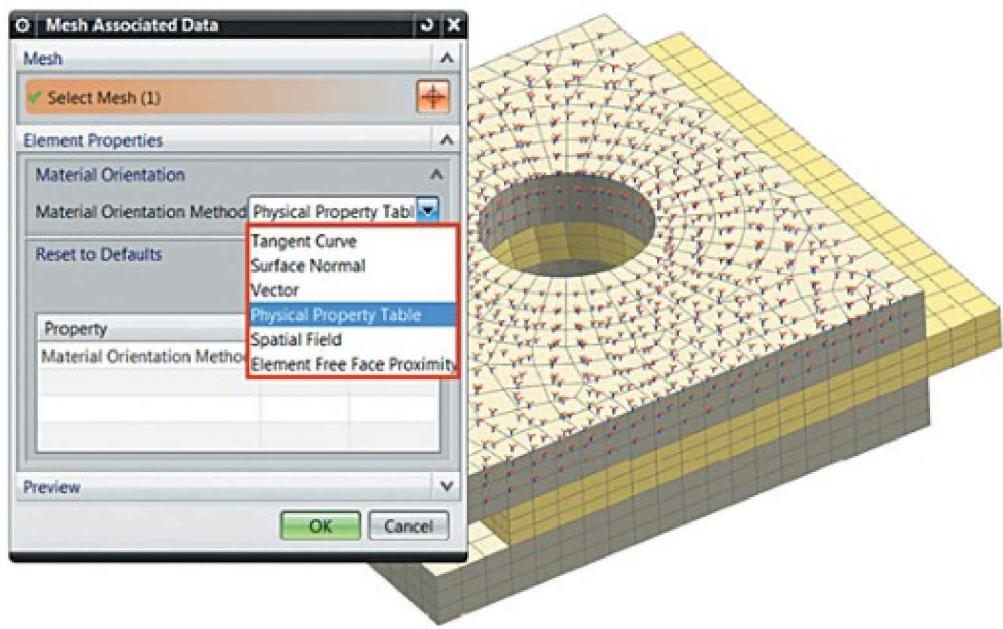


Figure 3.51. Associated data of 3D elements

For problems that involve shell-type 2D elements, a key aspect is the properties of these elements, thickness and offset, which are specified using the **Edit Mesh Associated Data** command. Here are the most important parameters of this command (Figure 3.52):

- Under *Select Mesh*, you can select the mesh for which you want to specify element associated data.
- In the *Shell Offset* box, you can enter the offset of elements relative to the surface on which the elements were constructed. Positive values correspond to the positive direction of the Z axis of the element coordinate system.
- Under *Material Orientation*, you can specify the orientation of material in one of the following ways: by referencing another coordinate system (*MCID*), by specifying the angle between the new and the old X axes (*Orient Angle*), by specifying X axis direction on *Tangent Curve*, or along a *Vector*.
- In the *Thickness Source* list you can select the method to use for

creating shell elements:

- *Physical Property Table*: thickness based on the specified value of the mesh collector (physical properties).
- *Midsurface*: thickness based on the thickness of the source geometry body if the elements are created on a midsurface. If you create a midsurface from a thin-shell body, it inherits thickness information. This thickness can be variable.
- *Field*: thickness varied in space in accordance following a specified formula or tabular values.
- Select *Preview* to preview the elements with the properties specified.

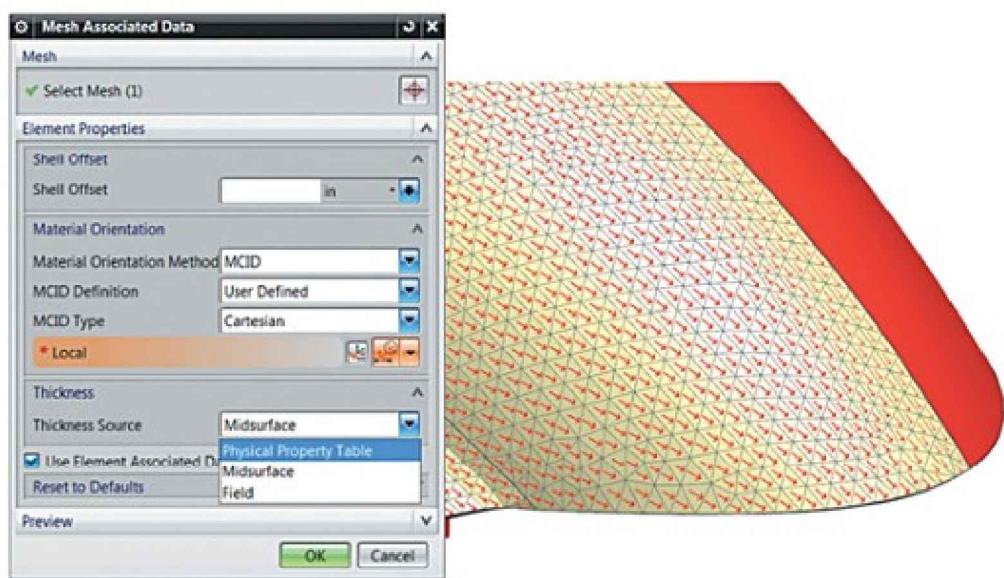


Figure 3.52. Associated data of shell elements

By default, shell elements are displayed with zero thickness. To verify that you set the thickness and offset values correctly, use the *Edit Display* command. Right-click the mesh or the mesh collector in the model tree view in the **Simulation Navigator** tab and choose the command as shown in Figure 3.53. You can use this command to select the color of element faces and edges, and visualize elements with actual thickness and directions of normal vectors.

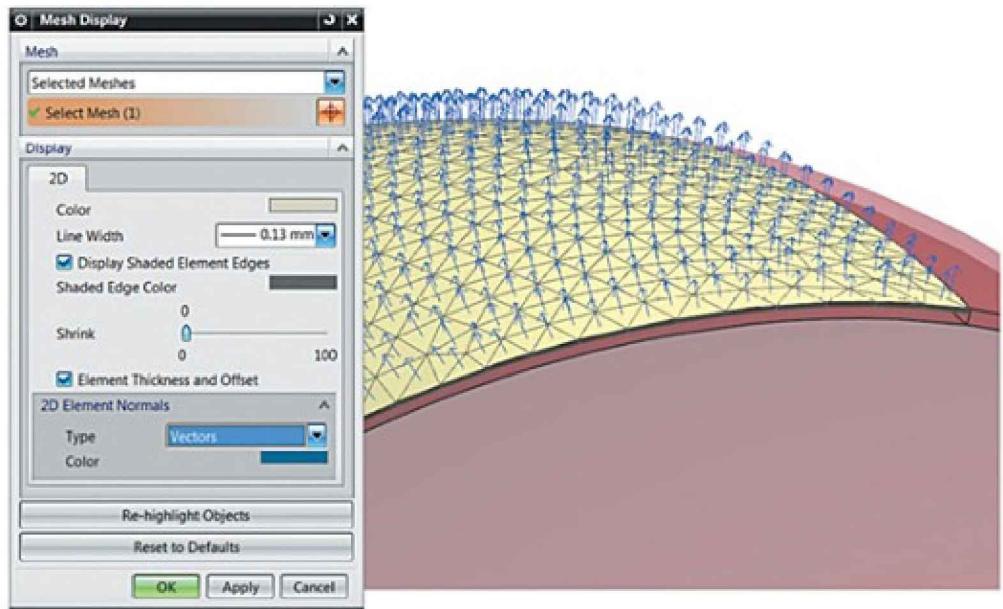


Figure 3.53. Visualization of shell elements

3.4.2. 0D and 1D elements. Working with beam elements

If the FE model contains 0D and 1D elements, it means they are used in modeling neglected parts of the structure with inertial, stiffness, and damping characteristics retained. Keep in mind that for some elements, the physical and related properties are defined not in the physical property collector but in the element associated data (Figure 3.54). You can call the corresponding **Edit Mesh Associated Data** command in the same way as for 3D and 2D elements.

To change the visualization of 0D and 1D, use the *Edit Display* command. Right-click the mesh or mesh collector in the model tree and choose this command.

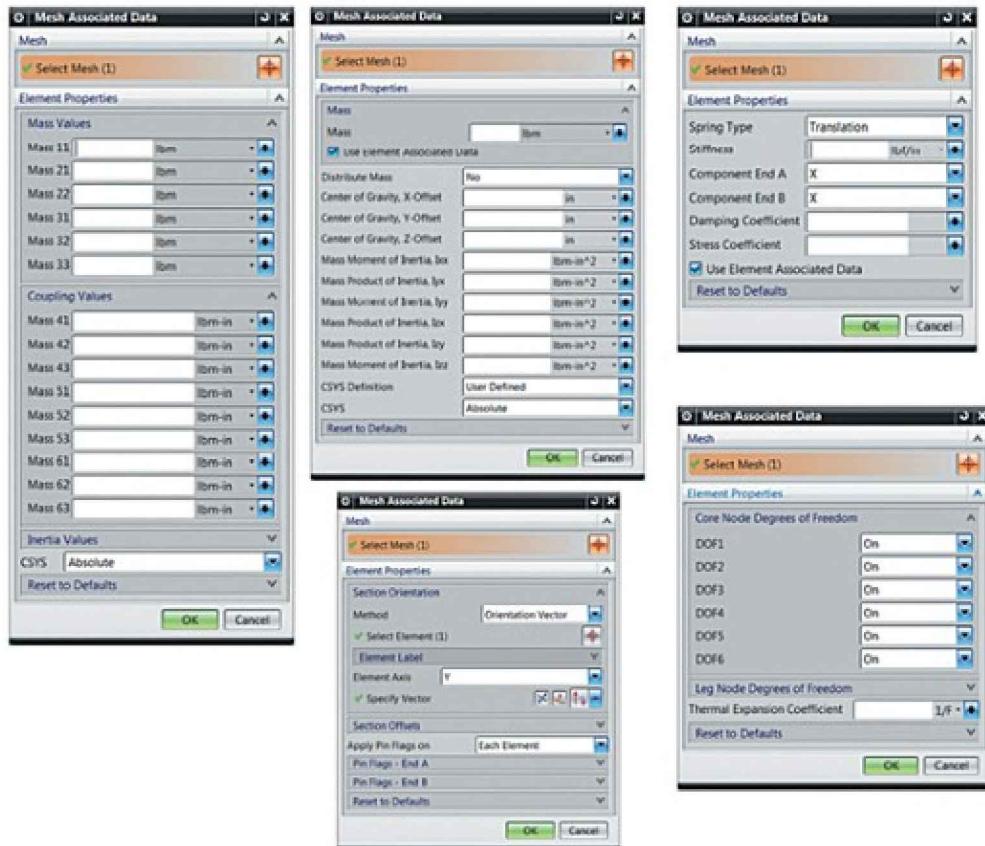


Figure 3.54. Associated data of some 0D and 1D elements

In **NX Advanced Simulation** simulation of frames using 1D is particularly prominent. The paragraph discussing the FE mesh creating tools describes creating rod elements based on curves and edges using the **1D Mesh** command, but there is more to this process. You need to specify the physical property table, and in addition to the material, specify the cross section created with the **Beam Section Manager** command. One possible application of mesh associated data is reorienting or offsetting a cross section. These stages of beam model creation merit a more detailed discussion.

You can open the **Beam Section Manager** dialog box from the physical property table or using the **1D Element Section** on the **Advanced Simulation** toolbar. The section manager contains all sections that were created in the FE model. You can display information concerning the dimensions and properties of the sections or simply visualize the relevant section (Figure 3.55). The most important features of the section manager

are creation of new sections and modification and deletion of existing sections.

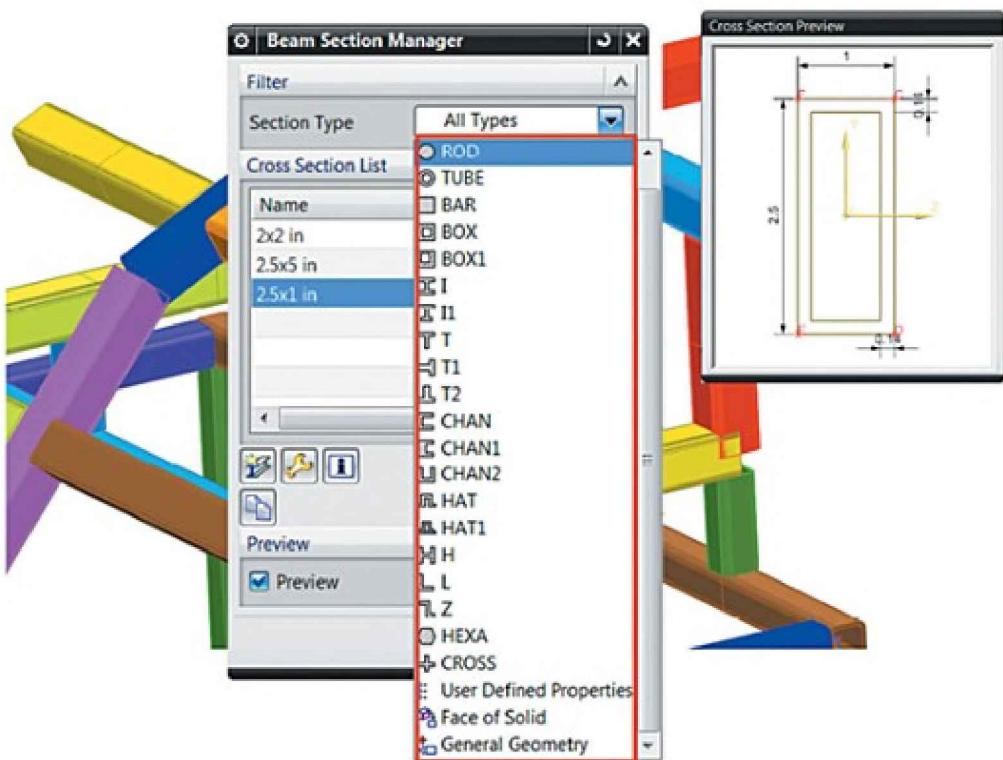


Figure 3.55. Section Manager

You can create a beam element using one of the following methods:

- You can use a section shape from the standard section library. In this case it is enough to specify the necessary dimensions (Figure 3.56, A).
- You can specify *User Defined Properties*. In this case you specify only the properties of the section but not its shape (Figure 3.56, B).
- You can select *Solid Face* to create arbitrarily-shaped sections based on a pre-created face of a body (Figure 3.56, C).
- You can use the *General Geometry* method to select one of pre-created sketches as the section shape (Figure 3.56, D). In this case you can simplify the task by using so-called layers, one of which contains the necessary sketch. After you select the sketch, you can hide the corresponding layer.

When you create a section, you can view its properties in a separate graphics window.

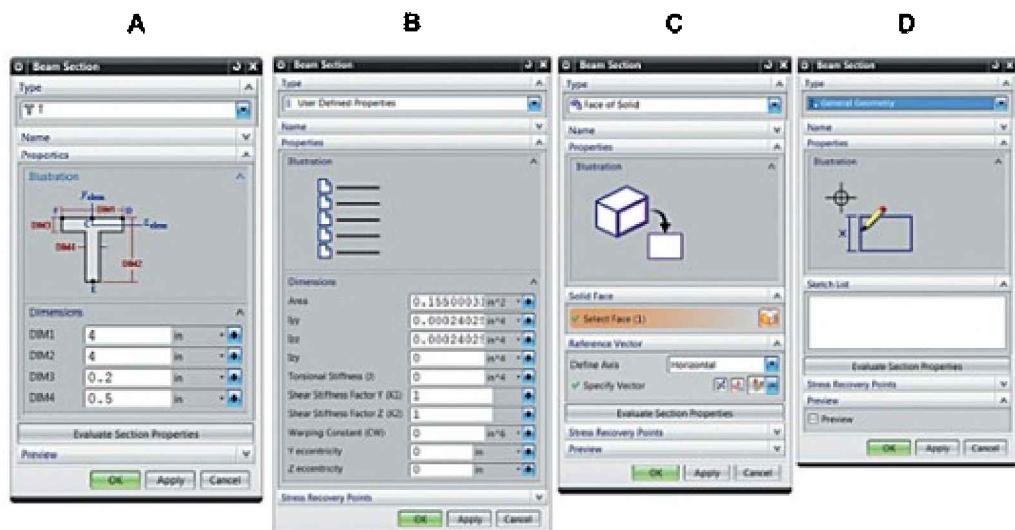


Figure 3.56. Creating the cross sections

An important aspect of creating beam elements is spatial orientation. A beam element has its own coordinate system where the OX axis is always directed along the geometry object that served as the basis of the element. Correct orientation of OY and OZ axes of the element coordinate system is typically established during section and mesh creation, but you can always reorient it using the **Edit Mesh Associated Data** command. You can use mesh associated data to specify a new orientation or offset a section with full control of active degrees of freedom of element nodes (Figure 3.57). Despite the apparent complexity of the mesh associative data dialog box, all its parameters are intuitive:

- In the *Section Orientation* group, you can select the spatial direction of one of the specified element axes (Figure 3.57, A).
- In the *Section Offset* you can set up section offset in one of three ways:
 - *Graphical:* a point on the section (Figure 3.57, B) is combined with a point belonging to another geometry object or with a point created in space using a shortcut menu. This method is especially effective if you need to describe a 3D body with beam finite elements.

- *Language-Specific*: the center of the section is combined with a selected point in space.
- Specify offset values relative to the element coordinate system.
- In the *Pin Flags* group, you can manage the degrees of freedom of element nodes by turning any of the six degrees of freedom on or off.
- In the *Reset to Defaults* group, you can restore parameters to a default state.

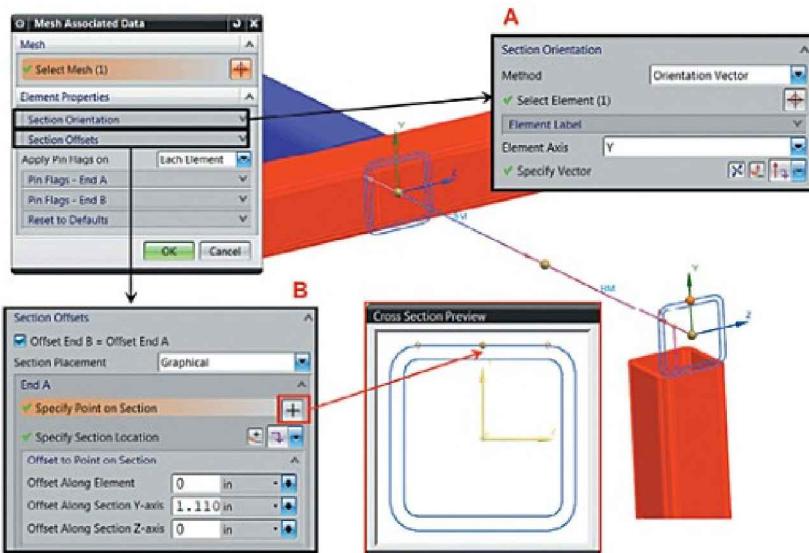


Figure 3.57. Associated data of beam elements

By default, beam element meshes are displayed as shaded objects that have the actual section shape. To change the display, use the *Edit Display* command. Right-click the mesh or the mesh collector in the model tree view in the **Simulation Navigator** tab. You can use this command to select element color, visualize elements with actual section in wireframe and solid representation, and display element coordinate systems (Figure 3.58).

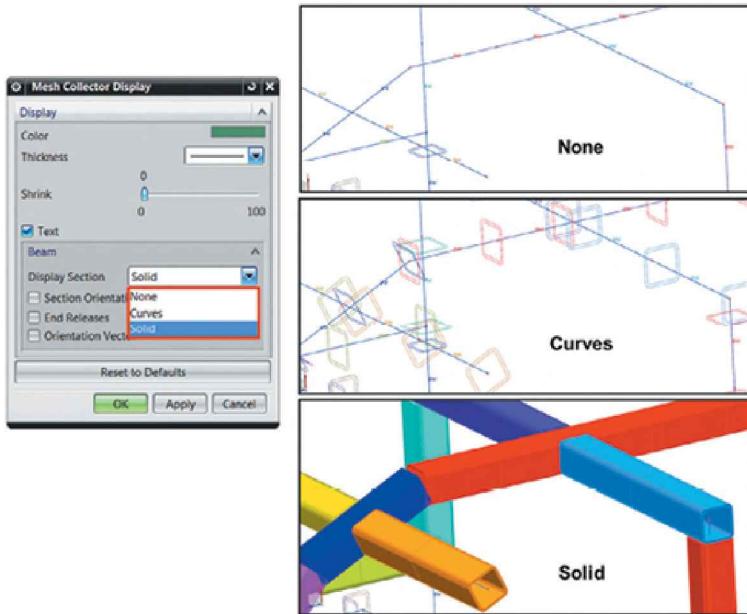


Figure 3.58. Visualization of beam elements

3.5. Additional features for FE models

3.5.1. Mesh parameters

When you create a mesh based on geometry, you specify general element size, but sometimes you need to manage element size and quantity on individual edges and faces of the object you generate the FE mesh for. You can split faces and edges using polygonal geometry operations. This is, however, a time-consuming process. You can save a substantial amount of time by using the ***Mesh Control*** command, which is available on the **Advanced Simulation** toolbar. This command creates splitting parameters on selected objects. These parameters appear in the model tree in the **Simulation Navigator** tab. When you use this command, you can select the method used to specify parameters and objects and set up options in accordance with the selected method. Here is a description of each type of splitting parameters with the relevant options:

- Select *Number on Edge* to specify the number of elements along the edge (Figure 3.59, A).
- Select *Size on Edge* to specify the size of elements on the whole edge, or on one end, or on both ends.
- Select *Chordal Tolerance on Edge* to specify chordal tolerance for

curvilinear objects to control the maximum distance between the curve and its chord (Figure 3.59, B).

- Select *Biassing on Edge* to specify the number of elements and the biasing factor for the selected biasing spot: start of edge, end of edge, or centre of edge.
- Select *Size on Face* to specify the size of element for selected faces.

You can preview the future node arrangement for the parameters you entered using a command in the *Preview* group in the same dialog bar.

The specified object splitting parameters are applied when you update the FE model or the corresponding mesh.

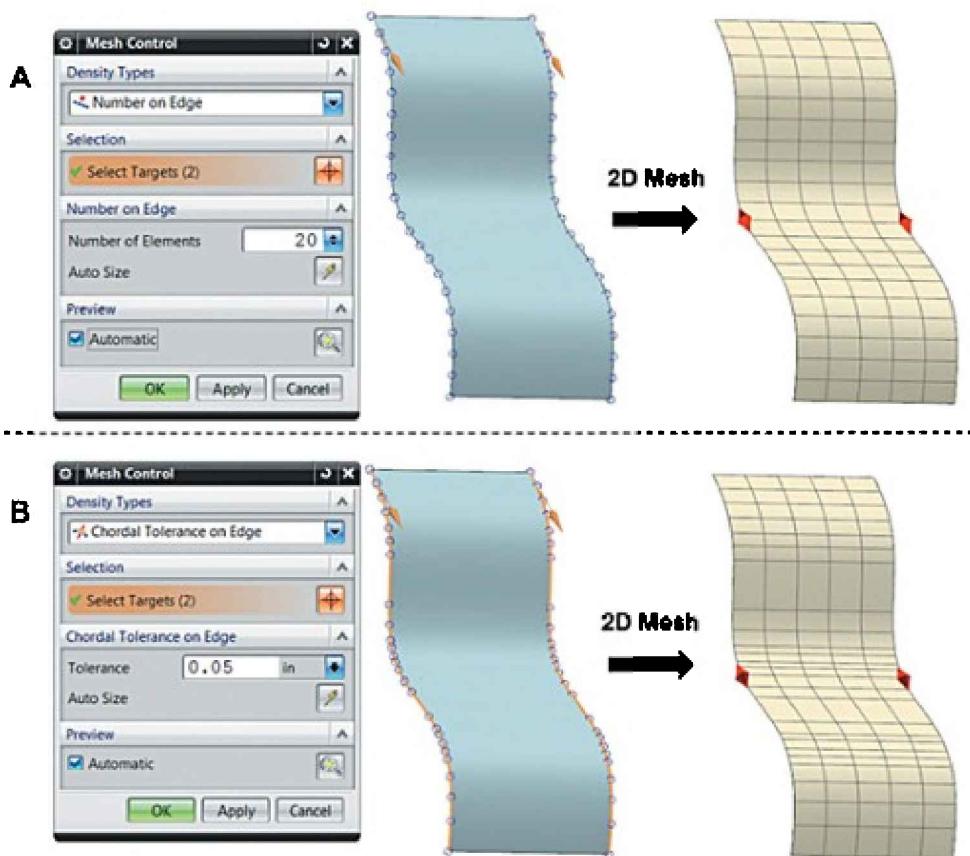


Figure 3.59. Mesh parameters

3.5.2. Checking your finite element model

After the finite-element model is created, you should perform some checks

to make sure there were no modelling errors:

- Display each collector separately and make sure it is correctly assigned to FE meshes.
- View each set of physical properties and mesh associated data, and make sure all parameters and values are set correctly. Pay special attention to use the same units.
- Perform some checks with the ***Finite Element Model Check*** command: check the shape of elements in the finite-element model, check for duplicate nodes and free element outlines, check normal vectors of shell elements, check material orientation of elements and the consistency of the FE model.

Here is a detailed description of the ***Finite Element Model Check*** command that is available on the **Advanced Simulation** toolbar. There are the following check types:

- Select *Element Shapes* to verify the shape of elements and find elements of unsatisfactory quality that can directly affect the result. This is one of the key checks of the FE model (Figure 3.60). The result is element diagnostic information in textual form. In the graphics area, bad elements are highlighted with a specified colour. There are several element shape distortion criteria: compression, deformation, skew, taper angle, Jacobian ratio, Jacobian zero, and 2D minimum/maximum corner angle. Each criterion has a limit value that can be modified. When “bad” elements are discovered, you need to rebuild the mesh or use ***Node Operations*** and ***Element Operations*** tool commands to change the shape of these elements.

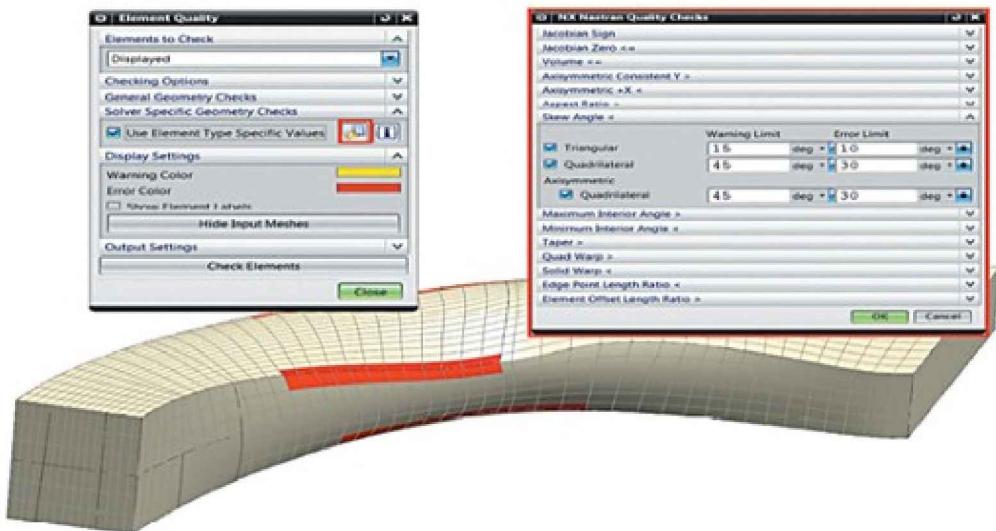


Figure 3.60. Checking element shape

- Select *Element Outlines* to check for free element outlines. As a result, all free outlines are highlighted (Figure 3.61). You can use this check to find elements within the model that are not connected to neighbour elements.

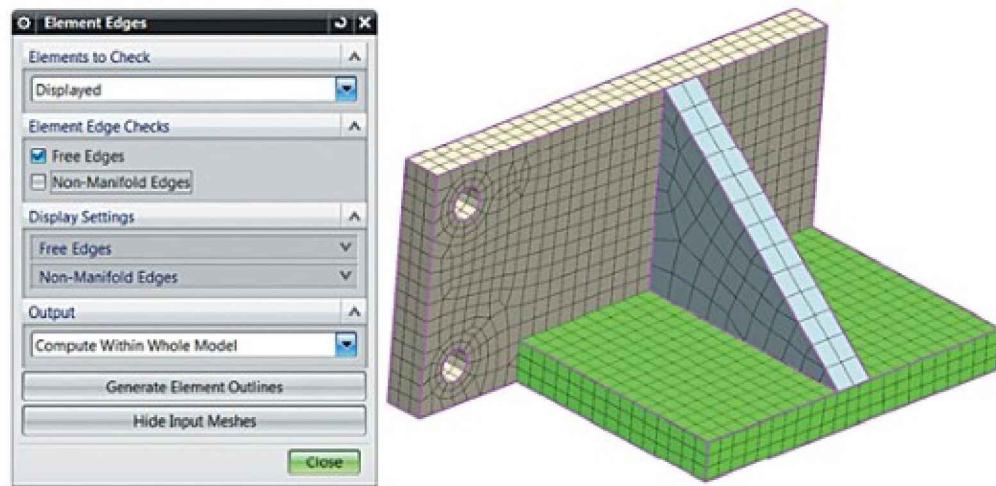


Figure 3.61. Checking free outlines of meshes

- Select *Nodes* to check for duplicate nodes with a specified tolerance. Such nodes indicate mesh discontinuity in these areas. To make the meshes continuous, you need to set the relevant mating conditions in the same dialog bar.

- Select *Element Duplicate* to check for duplicate elements.
- Select *2D Element Normals* to display normal vectors of 2D elements and modify them if necessary. You need to watch the direction of normals when you use an anisotropic material, specify shell element offset, work with multilayer shells, or specify pressure as the loading type.
- Select *Node Proximity to CAD Geometry* to find nodes that lie outside the geometry considering the given tolerance.
- Select *Element Material Orientation* to display the coordinate system of element materials.
- Select *CAE Model Consistency* to check the FE model for consistency. Find problematic polygon bodies.
- Select *Interference/Gap* to check for intersections and gaps between FE objects.

3.5.3. Importing/exporting FE models

Sometimes you need to work with a finite-element model that was not created in **NX Advanced Simulation** but in some other FE software system. Alternatively, you may encounter a case when you only have an FE model database in some solver format. In these cases you can import FE models in different formats. You can do this by choosing the **File→Import→Simulation...** command.

In the dialog box (Figure 3.62), select the solver type for which the FE model file was prepared, and click **OK**. In the new dialog box, set up some options depending on the solver type:

- In the *Input File Units* group, select the units of measurement that will be used to create a new FE model in **NX Advanced Simulation**.
- Select *File Type* to import (ASCII for text format, OP2 for binary format), and specify the file itself.
- In the *Selective Import* group, you can select individual objects for NX Nastran to avoid importing the whole FE model.

After committing the command, enter the name of the new FE model and the simulation file.

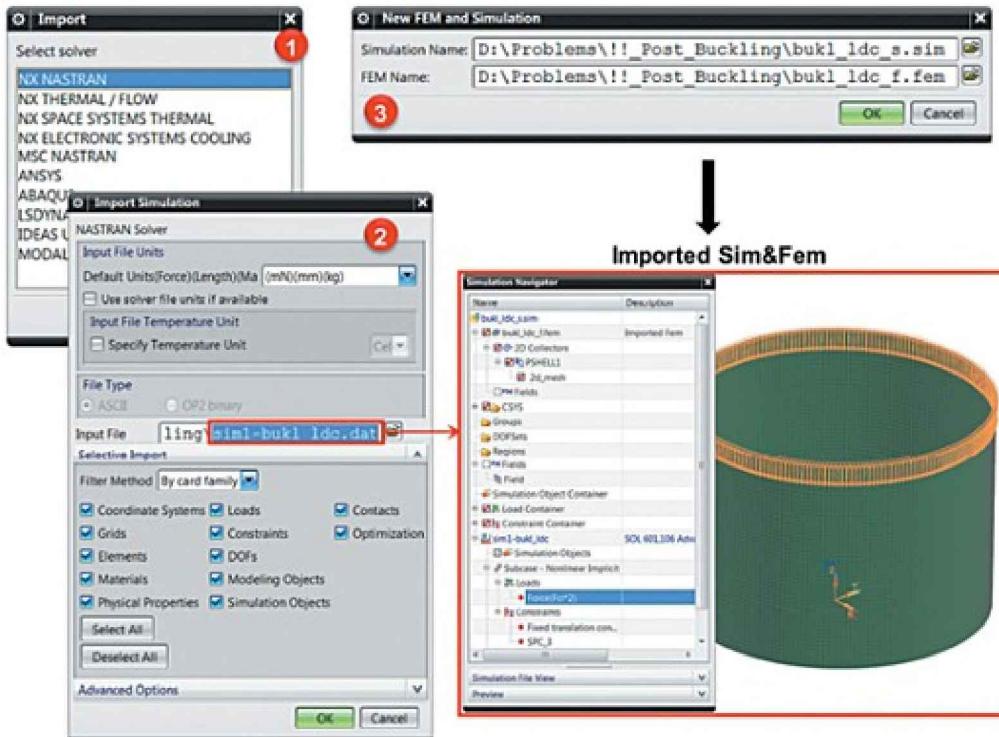


Figure 3.62. Importing a FE model

Note that the imported FE model does not contain geometry, but you can create polygon bodies based on element faces if necessary (**Face from Mesh** command) and use mesh-geometry association in subsequent operations.

NX Advanced Simulation has one more FE model import command. It is the **Append FEM** command on the **Advanced Simulation** toolbar. You can use it to insert any other NX format model into the current FE model (Figure 3.63). To do so, select one of the FE models loaded into NX in the command dialog box, and specify the number offset value for renumbering of nodes, elements, and physical properties.

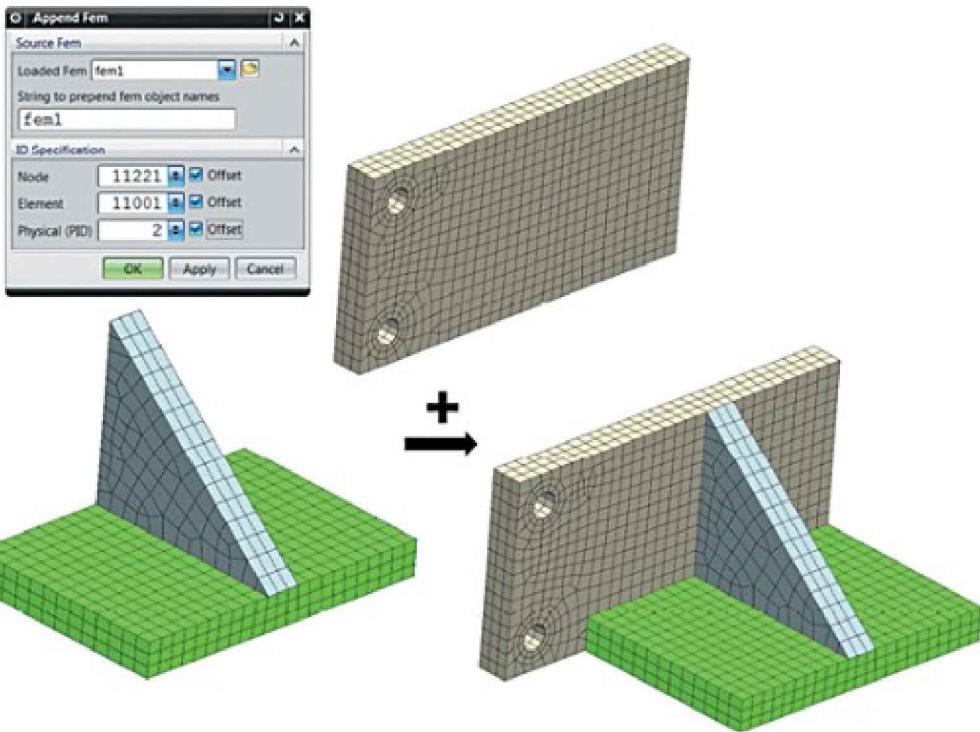


Figure 3.63. Adding a FE model

You can export an FE model from NX Advanced Simulation as a file in the format of the solver that was used to create it, in the **File→Export→Simulation...** dialog box, where you need to set up the following options (Figure 3.64):

- Under *Output File Units*, select the units of measurement to use in the output file. Values specified in NX are converted into these units.
- In the *Output File* box, enter the name and location of the export file.
- In the *Subset Export* group, you can select a part of the FE model for export.
- In the *Output Options* group you can select FE objects for export.
- In the *Entity ID Offsets* group, you can specify the numbering offset values for renumbering of FE objects, and select the number of the coordinate system in which the FE model is exported.

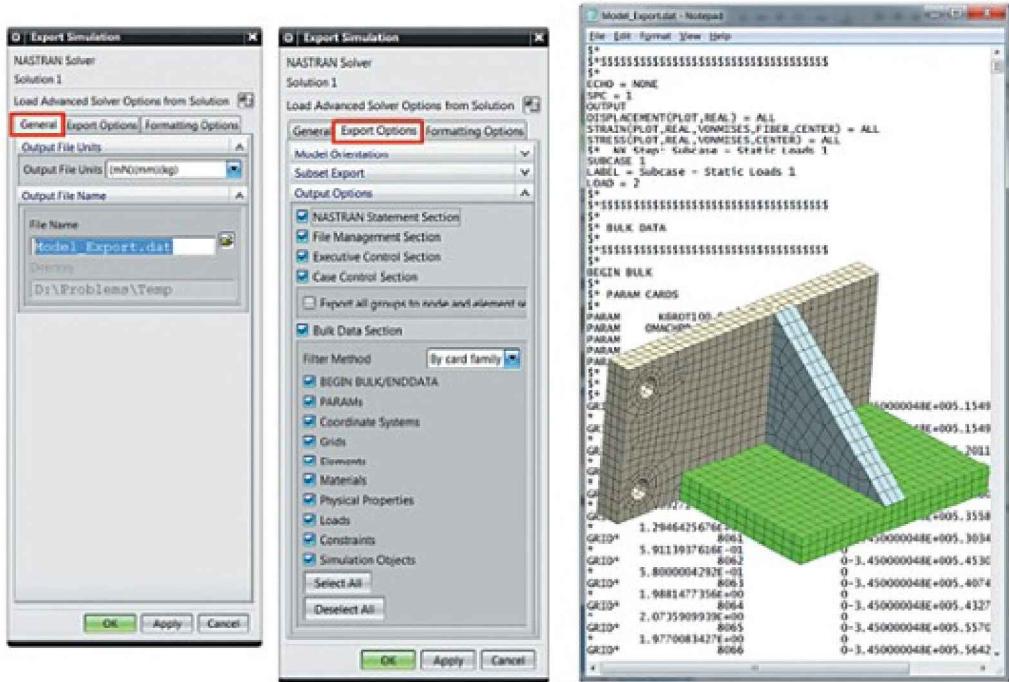


Figure 3.64. Exporting a FE model

3.6. Working with polygon geometry

Sometimes you will encounter a situation when a geometry model is too detailed for numerical simulation. When you work with the FE model, you can correct the polygon geometry without transitioning to the idealized model to improve quality and manage the FE mesh. There are special tools, the so-called abstraction tools, which you can use to remove degenerate faces and extreme surfaces (geometry healing), split and join faces and edges, fill in gaps, and restore missing faces. These commands do not modify the idealized geometry model. A critical aspect of working with geometry editing tools is the ability of NX Advanced FEM pre-processor to automatically discover all modifications and update the finite-element model. The loads, boundary conditions and other simulation options are retained.

You can access polygonal geometry commands in one of the following ways:

- Choose **Insert→Model Cleanup** in the main menu.
- Click the command on the **Advanced Simulation** toolbar.
- Use commands on the **Model Cleanup** toolbar.

Let us describe the abstraction tools (Figure 3.5) and provide detailed information on the most useful commands for FE mesh building. When you perform any operations, keep in mind the point selection references, which can significantly simplify selection of points and prevent selection mistakes. With some commands, you need to commit the current polygon geometry operation by clicking **OK** or **Apply** even if you can see the result in the graphics area. All abstraction commands can be used both before and after FE mesh creation. If you use abstraction after building the mesh, you can simply update the FE model using the **Update Finite Element Model** command in the **Advanced Simulation** command.

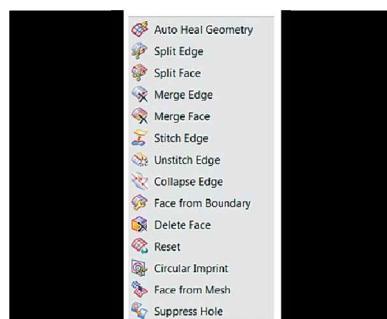


Figure 3.65. Abstraction tools

Use **Auto Heal Geometry** to automatically discover and heal problematic areas of polygon geometry. Select polygonal objects and enter the size of small geometry elements to remove.

Use **Split Edge** to split edges using one of three methods:

- Select *Location on Edge* to specify point location on an edge.
- Select *Project Point on Edge* to specify the edge for splitting and the point whose projection along the normal vector of the edge divides the edge into parts.
- Select *Project Point Along a Vector* to specify the edge for splitting and the point whose projection along the given vector divides the edge into parts.

Select **Split Face** to split faces into parts in one of the two ways:

- Select *Split face by points* to specify two points, one per edge of the face.
- Select *Split face by suppressed edges* to highlight edges of the

polygonal geometry that existed before the **Merge Face** abstraction operation. Then select the edges that you want to use for face splitting (Figure 3.66). This command substantially simplifies the creation of a high-quality FE mesh and boundary condition setup in localized areas.

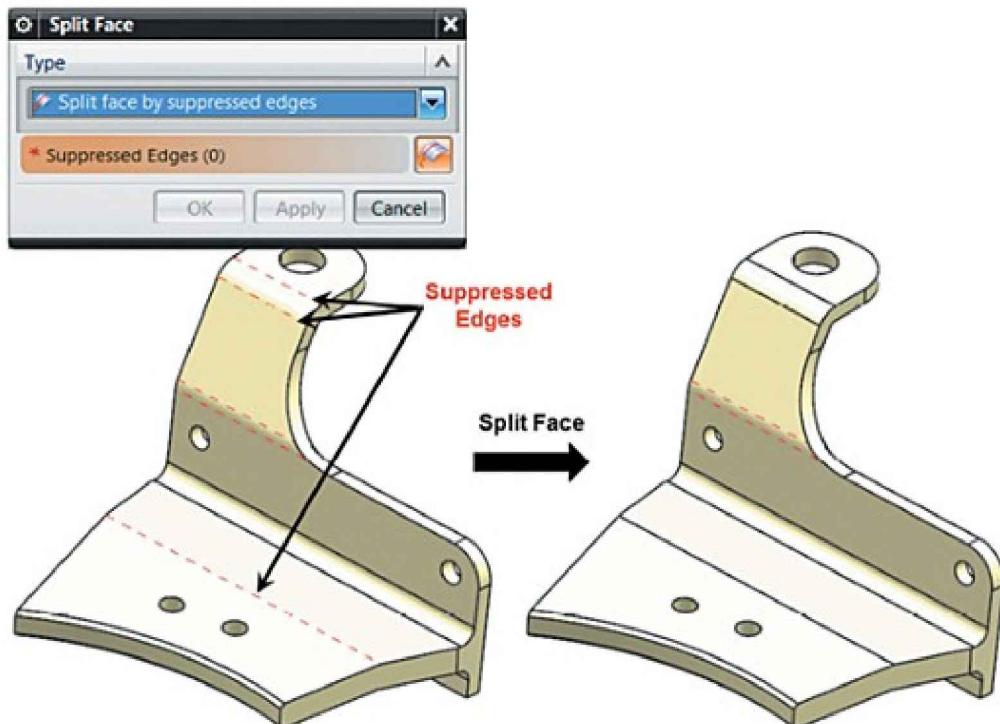


Figure 3.66. Splitting faces

Select **Merge Edge** to merge two edges that have a shared point and belong to the same face. When you use this command on the polygon geometry, shared points of edges that can be merged are highlighted.

Select **Merge Face** to merge adjacent faces of the same body. You can select the faces using one of the two methods: selecting the shared edge of adjacent faces or selecting several faces for merging. You can use this command to create a high-quality FE mesh by removing small faces through merging them. It enables and simplifies the creation of structured hexahedral FE mesh (Figure 3.67).

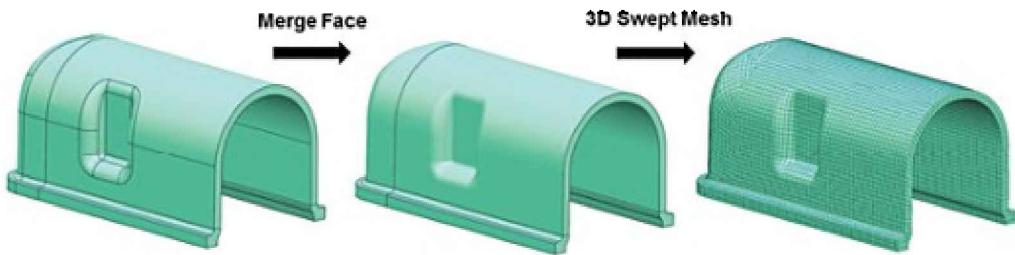


Figure 3.67. Merging faces

Select **Stitch Edge** to perform edge-to-edge and edge-to-face stitching with removal of any gaps. When you open the corresponding dialog box, all free boundaries of polygon faces are automatically highlighted in the graphics area to let you perform a visual search of problematic areas. The command is useful when you need to work with midsurfaces and faces to create a continuous shell mesh (Figure 3.68). You can use this operation to automatically find stitchable objects among the specified set of polygon bodies and faces. Alternatively, you can specify the objects for stitching manually. This command is convenient because it is saved in the model tree so it is automatically performed when polygon geometry is updated if geometry is modified as part of the idealized model.

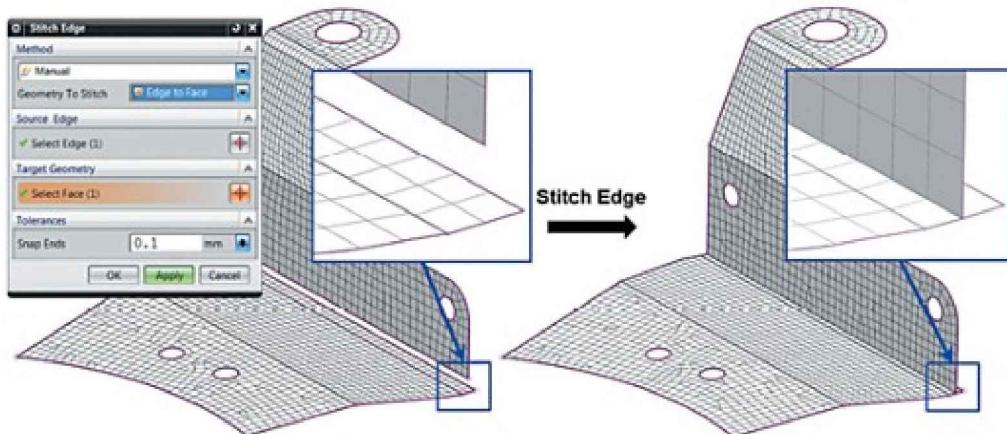


Figure 3.68. Stitching edges and faces

Select **Unstitch Edge** to reverse a **Stitch Edge** operation but without restoring any gaps that might have existed before face stitching.

Select **Collapse Edge** to remove edges, replacing them with a specified point and rebuilding adjacent faces (Figure 3.69).

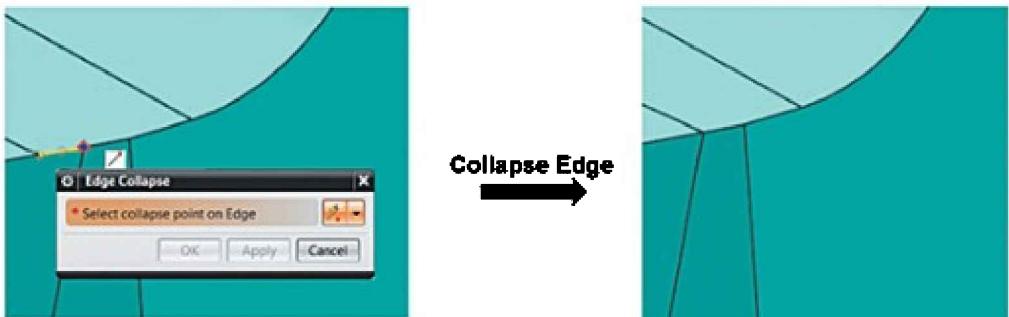


Figure 3.69. Removing edges

Select **Face Repair** to repair missing faces or remove problematic faces. This command is useful when there are missing faces as a result of incorrect geometry import or when there are small objects.

Select **Reset** to remove all performed abstraction commands for selected polygon objects.

Select **Circular Imprint** to create a circular area around selected circular edges or points with the specified diameter (Figure 3.70). Depending on the object selection, the method used to specify the diameter of the new face can vary. Here is a description of the most important options of this command:

- Use *Select Surface* to specify a face on which to create the circular area.
- Use *Select Point or Edge* to specify points or circular edges of a selected face. The circular area is created relative to these points or edges.
- Specify the diameter of the new circular areas in the *Diameter of Imprint* group:
 - Select *Around Point* to enter an absolute diameter value for the new areas in case you select points.
 - Select *Around Edge* to enter either the diameter value for the new area or the scaling factor for diameters of the source circular edges.

This command substantially simplifies creation of high-quality regular FE meshes around holes.

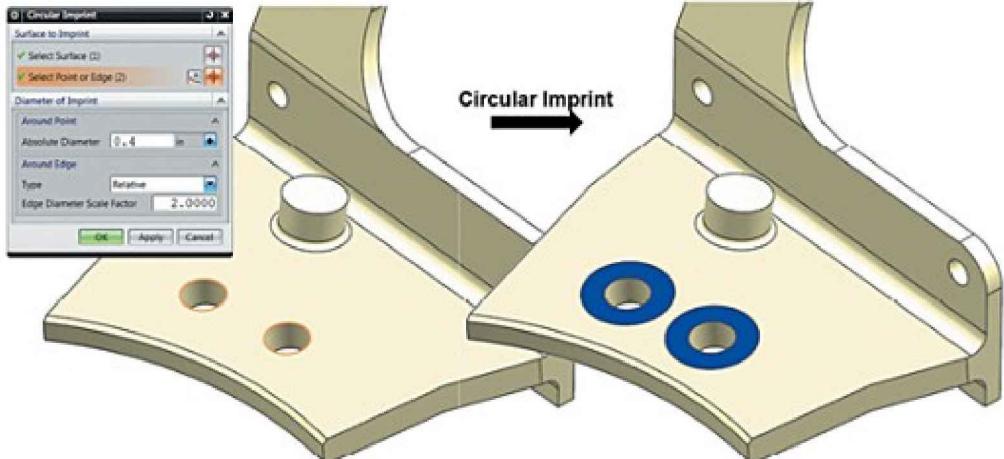


Figure 3.70. Creating a circular imprint

Select **Face from Mesh** to create polygon geometry based on the selected faces of 2D finite elements or free faces of 3D elements. As a result of this command, a new polygon face is created, and further mesh-based faces can be added (Figure 3.71). The new polygon object remains a set of only faces until these faces form an enclosed area. In this event it is transformed into a polygon body. You can use this command to create polygon geometry based on an imported or created FE model that lacks geometry. For example, you can establish association of finite-element and geometry objects to simplify subsequent processing of FE meshes, loads and boundary conditions.

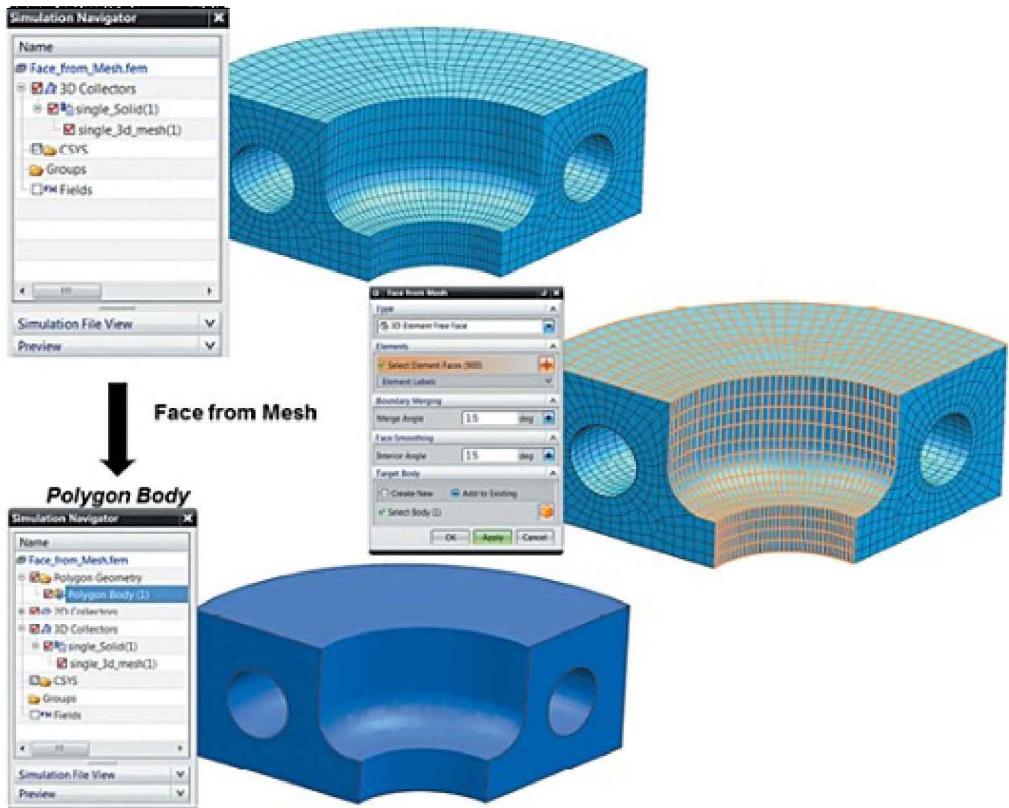


Figure 3.71. Creating polygon geometry

Select **Suppress Holes** to create circular areas instead of selected holes and join them together with the primary face. Therefore, this command results in suppression of holes on faces (Figure 3.72). Any holes, not just circular ones, can be subjected to suppression. You can automatically select all holes on selected faces that have diameters less than or equal to the entered value. Alternatively, you can specify the relevant edges manually. You can automatically create geometry points and mesh points at the centres of removed holes. You can use this command to quickly (and reversibly) delete holes that do not influence FE simulation results but introduce a large number of extra finite elements.

This operation is saved in the model tree, so the polygon geometry and the FE mesh can be updated automatically whenever the idealized model is modified.

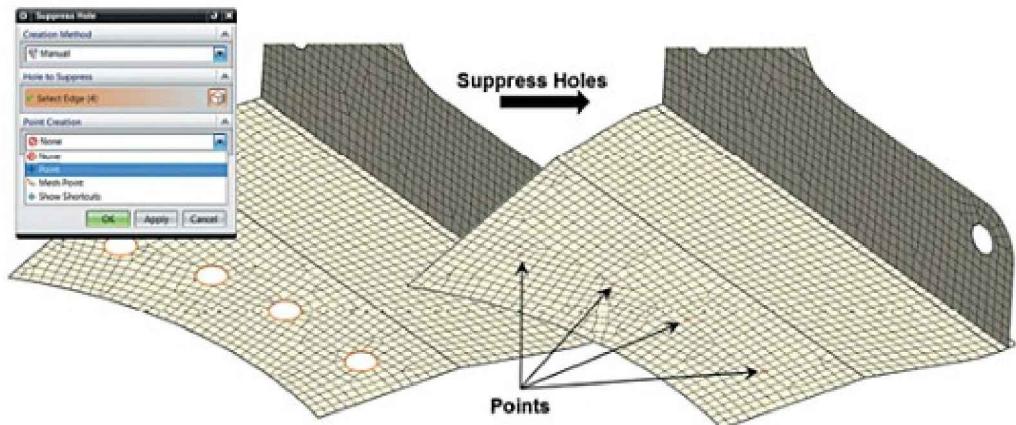


Figure 3.72. Suppressing holes

3.7. FE models of assemblies

Quite often you need to use CAD assemblies consisting of a great number of parts as the source geometry model for finite-element analysis. The CAD assembly typically contains identical parts represented by a single geometry file. When you create a standard FE model based on the source assembly, you need to consider not just a set of individual parts, but a multitude of bodies, therefore you need to generate a mesh for each polygon body used in the FE model (Figure 3.73, A). This method is adequate for small structures but not for complex and cumbersome assemblies.

As an alternative to creating an FE model from the source assembly, you can create a so-called *Assembly FEM*, which is similar to an assembly geometry model in that identical parts are represented by a single FE model and therefore a single file (Figure 3.73, B). Working with an FE assembly is similar in principle to working with its CAD counterpart. You can add and remove FE components, modify mesh positions, and create other finite elements to link components (meshes). One assembly FE model can be processed by many engineers, each of them responsible for particular components of the assembly FE model. If necessary, you can use the same assembly part FE model in different assembly FE models. This procedure is also supported by the Product Lifecycle Management (PLM) system, Teamcenter, through a dedicated module called Teamcenter Simulation Process Management

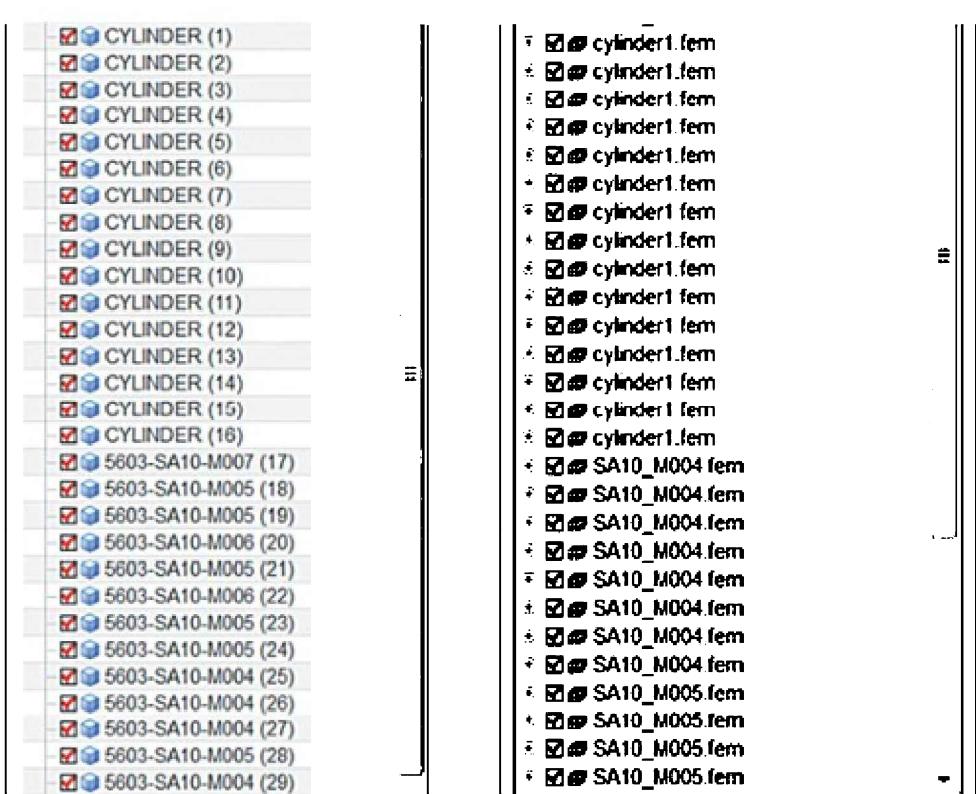


Figure 3.73. FE model tree and assembly FE models

3.7.1. Cloning assembly FE models

Depending on the source data and requirements, you can use one of the two assembly FE model creation methods:

- Associative, which involves creating an assembly FE model based on the CAD assembly model and adding new or existing FE models of individual components. Therefore you retain full association of assembly FE models and parts with their CAD counterparts.
 - Non-associative, which involves creating an empty assembly FE model and populating it with FE models by specifying their location and spatial orientation.

The following is a step-by-step description of the assembly FE model creation process using the more widespread associative method, using a

specific structure as the example (Figure 3.74).

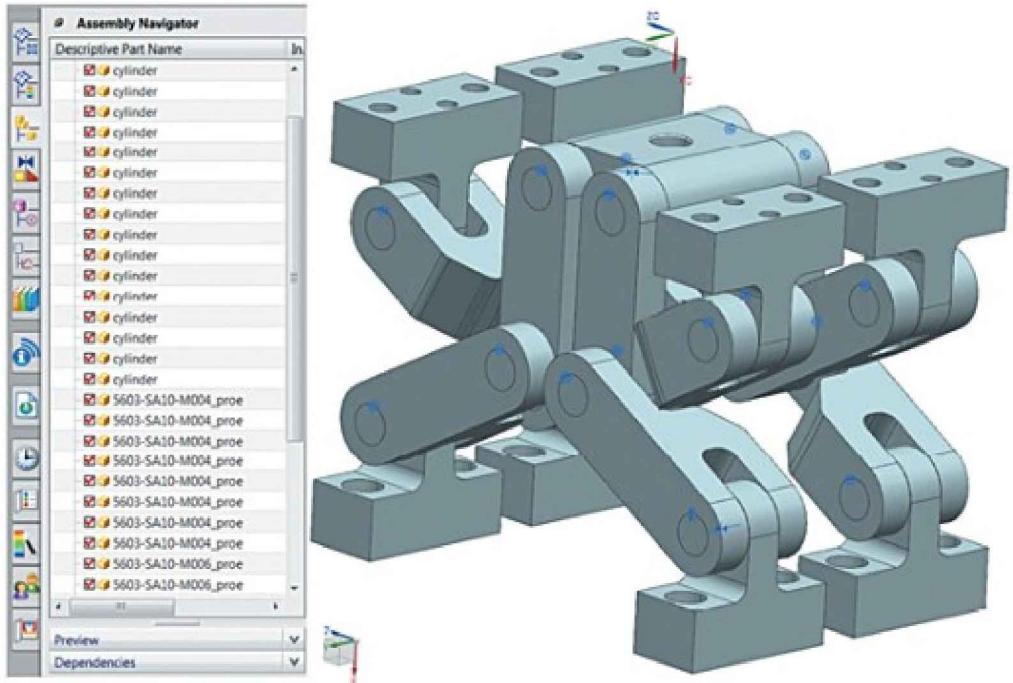


Figure 3.74. Assembly CAD model

1. Launch NX and open the *assembly1.prt* file.

This file corresponds to the assembly CAD model of the structure in question, which contains eight identical *5603-SA10-M004_poe* parts, eight *5603-SA10-M005_poe* parts, four *5603-SA10-M006_poe* parts, one *5603-SA10-M007_poe* part and sixteen *cylinder* parts (Figure 3.74).

2. Go to NX Advanced Simulation (*Start→Advanced Simulation*). Set default options for all dialog boxes. To do this, choose *Preferences→User Interface→General→Reset Dialog Box Settings* in the main menu, then click *OK*.

Right-click the assembly CAD model *assembly1* in the **Simulation Navigator** or the **Simulation File View** dialog box (Figure 3.75, A). Select **New Assembly FEM** to create the assembly FE model.

In the assembly FE model creation dialog box, enter the name and path for the new file (Figure 3.75, B), then click *OK*. In the new dialog box (Figure 3.75, C), you can preview the source CAD assembly, and specify the solver and analysis type. At this stage all setting retain default values, so just click

OK.

3. You created an assembly FE model based on the original CAD model. The new assembly FEM is initially empty, that is, it does not contain any FE models of assembly components.

In the **Simulation File View** dialog box, the assembly FE file is active. Note that in the model tree of **Simulation Navigator** contains a list of CAD assembly components with crossed-out icons (Figure 3.75, D), which means that these components currently have no FE models defined.

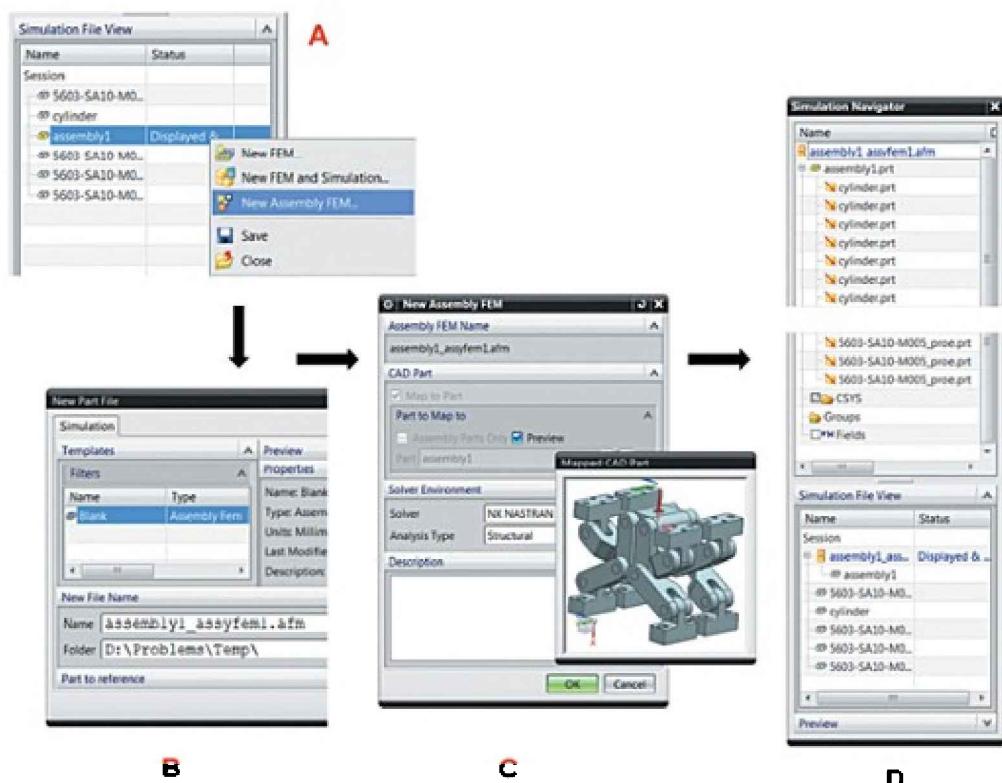


Figure 3.75. Creating an empty assembly FEM

The next step is creating FE models of assembly components based on the corresponding CAD parts. This means that each unique part undergoes a standard FE model design process with mesh creation and physical property definition. The FE models you create are always associated with the corresponding geometry models. Omit this stage and use existing FE models shown in Figure 3.76.

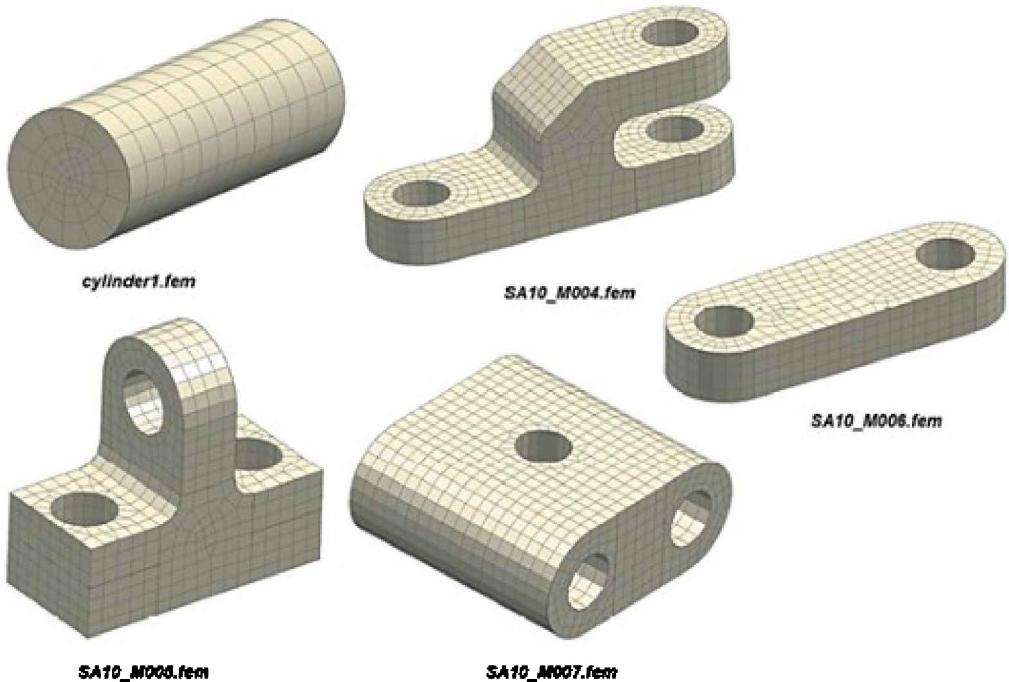


Figure 3.76. FE models of assembly components

4. Choose **File→Open** in the main menu to load an FE model file for each unique component: *SA10-M004.fem*, *SA10-M005.fem*, *SA10-M006.fem*, *SA10-M007.fem*, *cylinder1.fem*. This FE model opening step is not mandatory but it simplifies the subsequent creation of assembly FE model.
5. Note that the **Simulation File View** dialog box shows all loaded files, and the last loaded FE model file is active. Double-click the assembly FE model file (Figure 3.77, A).

The next step involves adding component FE models one-by-one to the assembly FE model. The order is not important. In the assembly FE model, right-click *5603-SA10-M007_proe.prt* and select **Map Existing....** In the dialog box, select the FE model associated with the selected CAD model of the assembly component (Figure 3.77, B). There are three ways to do this:

- Use the *Open* command to select the FE model file in the corresponding folder.
- Select the model in the *FE Model* list. This list contains all currently loaded FE model files.
- Select the *List Only Associated Models* checkbox. If in addition to

the FE model, the corresponding CAD model is also loaded, the *FE Model* list contains only the associated FE model. This method is most convenient but it works only if the FE model file and the CAD model file are both loaded.

Because the CAD model is not loaded yet, follow the second method. Select SA10-M007 in the list and click **OK**. In the graphics area you can see the FE model that was added to the assembly FE model, and the tree view shows the corresponding CAD component highlighted in green.

If the assembly consists of identical components that correspond to a single FE model, you don't need to repeat the previous step for each of the CAD components. You only need to select all identical ones and add the corresponding FE model. In the FE model tree, right-click *cylinder.prt* and choose the **Find All Matching Components** command (Figure 3.77, C). For highlighted components, right-click and choose the **Map Existing...** command. In the dialog box, select the *cylinder1* FE model associated with the selected CAD model of the assembly component. **Click OK**.

Repeat this step to add the rest of the FE models for *5603-SA10-M004_proe.prt*, *5603-SA10-M005_proe.prt*, and *5603-SA10-M006_proe.prt* components.

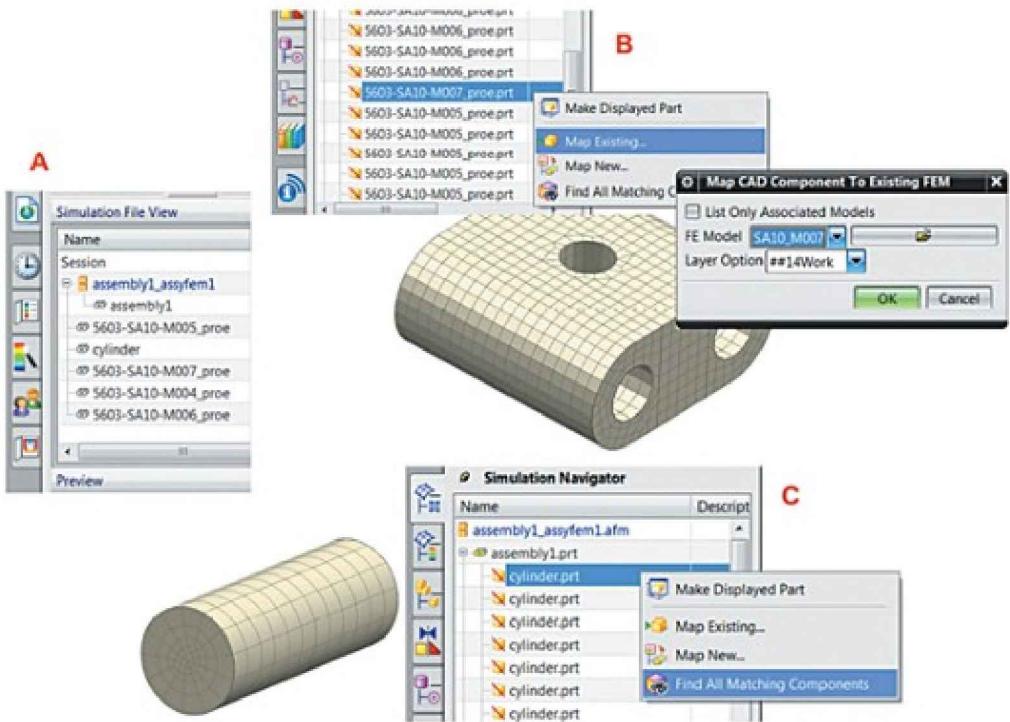


Figure 3.77. Populating the assembly FE model

After populating the FE model, you get a set of component FE models arranged in space to reflect the relative arrangement of CAD components (Figure 3.78).

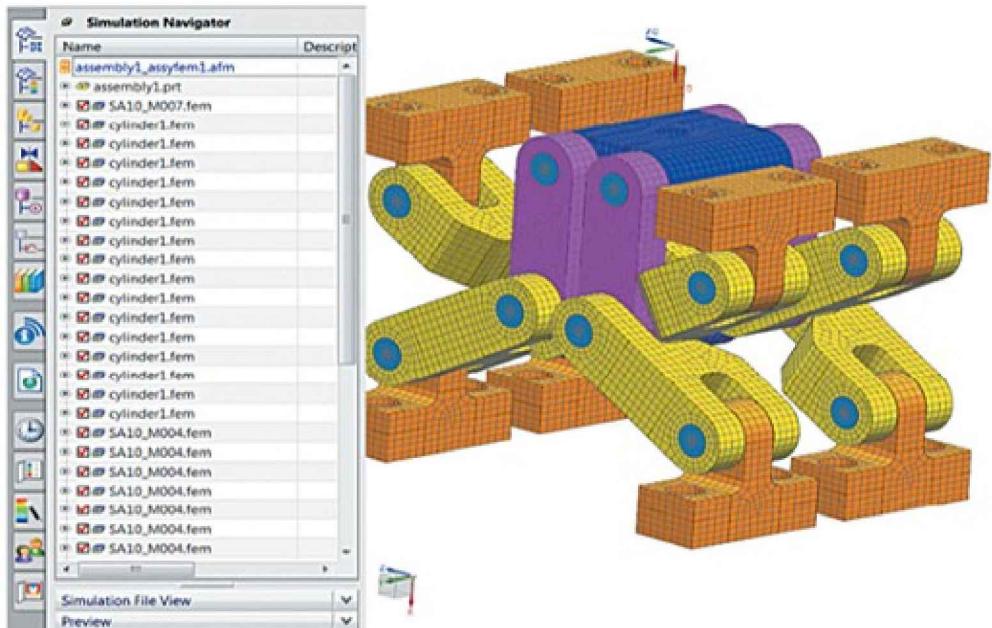


Figure 3.78. Assembly FEM

If you use the non-associative method of creating the assembly FE model, the only way it differs from the associative method is the populating of the empty assembly FE model with FE models without the source CAD assembly. Therefore, you need to specify the location and spatial orientation of FE models you add.

To finish creating the simulation model, complete the following steps: resolve numbering conflicts, join FE components, create the simulation model (Simulation Part) with relevant boundary conditions and interaction conditions of FE assembly components. These steps are discussed in greater detail in the following section.

3.7.2. Special aspects of working with assembly FE models

After you create an assembly FE model consisting of component FE models, you typically encounter a node, element, and coordinate system numbering conflict. Each FE model of each component is created separately, therefore there is some overlap in the numbering of finite-element objects. This is unacceptable because each node, element, and coordinate system within a unified FE model must have a unique number. To resolve this kind of conflict, choose the **Assembly Label Manager** command in the shortcut

menu that you can access by right-clicking the assembly FE model node (Figure 3.79) in the model tree view of the **Simulation Navigator**. In the dialog box that opens, conflicting objects are marked with red crosses. You can renumber the objects manually or automatically by specifying a numbering offset. Here are the most important parameters of the numbering conflict resolution command:

- Under *Type* specify the offset selection type.
- Enter *Offset to Nearest* to specify the order of offset value.
- Use the *Automatically Resolve* command to automatically renumber objects. This command is used in most cases, except those cases when you need the numbering to follow a certain rule.
- In the *Labels* group you can display conflicting objects for each of the three types, and manually specify offset values for each FE model.

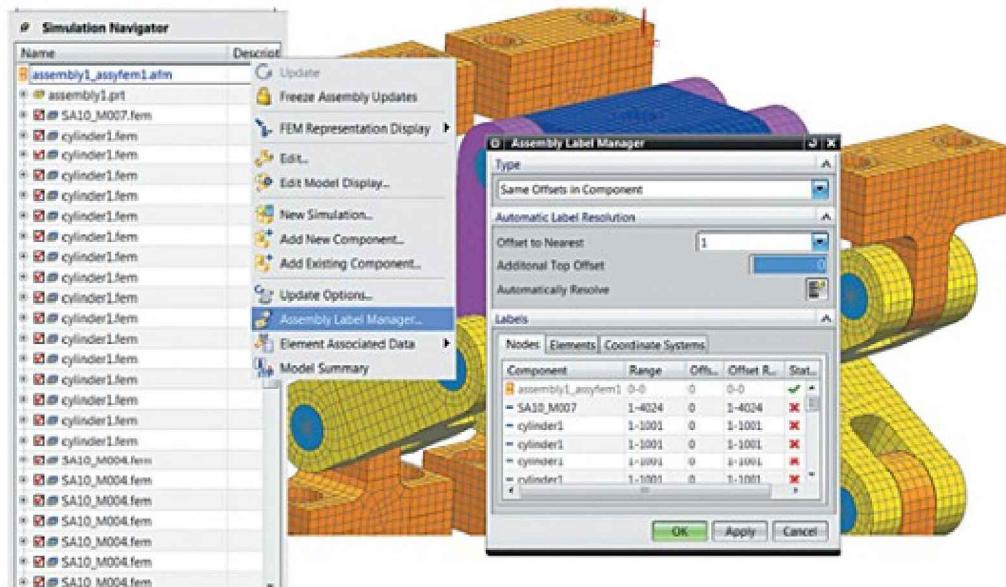


Figure 3.79. Managing numbering

After you create assembly FE model, you need to connect the separate FE components into a unified FE model. This can be achieved using some of the tools mentioned earlier (Figure 3.80):

- Select **1D Connection** to connect the node set of an FE component with another node set through various 1D elements,

for example, to specify stiff and elastic connections to correctly transmit forces and displacements.

- Select **Bolt Connection** to model bolt connections that join together individual parts of the structure.
- Select **CFAST/CWELD Connection** to create special NX Nastran CFAST/CWELD connections between the FE meshes. You can use connections of this type to model riveted and welded structural elements.
- Select node and element operation commands that you can use to create new finite elements for connecting FE components or as additional objects for the common FE model.
- Select **Finite Element Model Check** to merge duplicate nodes in the area of touching FE components.

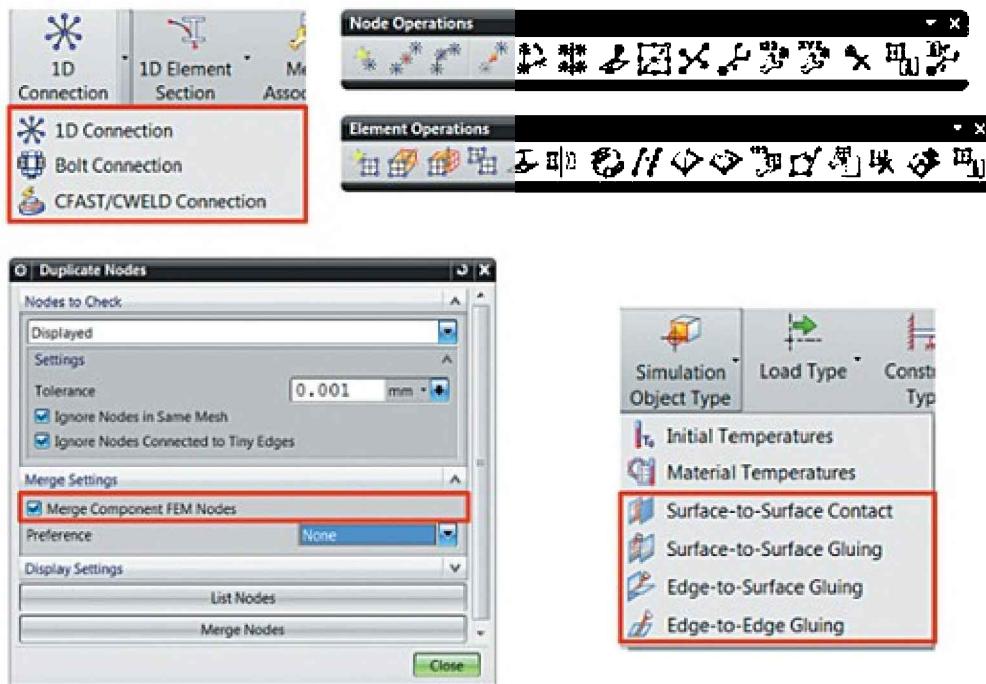


Figure 3.80. FE component creation tools

In addition to creating connection finite elements, you can use **Simulation Objects** to model correct interaction of FE components in the simulation model (see Chapter 4), such as:

- **Surface-to-Surface Contact**, which allows to set contact conditions for two parts of a structure.

- *Surface-to-Surface Gluing* and *Edge-to-Surface Gluing*, which models the lack of relative movement of selected areas and correctly transmits forces and moments.

An indisputable advantage of using assembly FE models is complete association with source CAD assemblies. This allows updating the resultant assembly FE model whenever the CAD assembly is modified (at assembly level), for example, if relative positions of components change. Keep in mind that if you modify the CAD model of a component, you need to update the FE model of this component and then renumber the finite-element objects in the assembly FE model.

When you use NX Nastran solver and replace some parts of the structure with so-called „super elements“, you can replace some assembly FE model components with external super elements. Specify a result file (**_0.op2*) for the component or identical components being replaced. This file is created when you run “SOL101 – Superelement” or “SOL103 – Superelement” NX Nastran analyses.

In this way, using super elements can substantially reduce the number of equations for large assemblies.

To use super elements in assembly FE models, perform the following steps (Figure 3.81):

1. In the completed assembly FE model, right-click a FE component in the model tree view. Choose the **Edit Attributes** in the shortcut menu.
2. In the **FE Model Occurrence Attributes** dialog box, under *Representation*, select *Super Element*.
3. In the *Super Element File*, select the location of the file that contains the results of super element analysis performed in advance.
4. In the *Graphical Representation* group, select super element display mode for the assembly FE model:
 - Select *None* to display only boundary nodes.
 - Select *Approximation of base FEM* to display only the dummy FE source component.
 - Select *Super Element Symbol* to display super elements and boundary nodes as symbols.
5. Click **OK**.

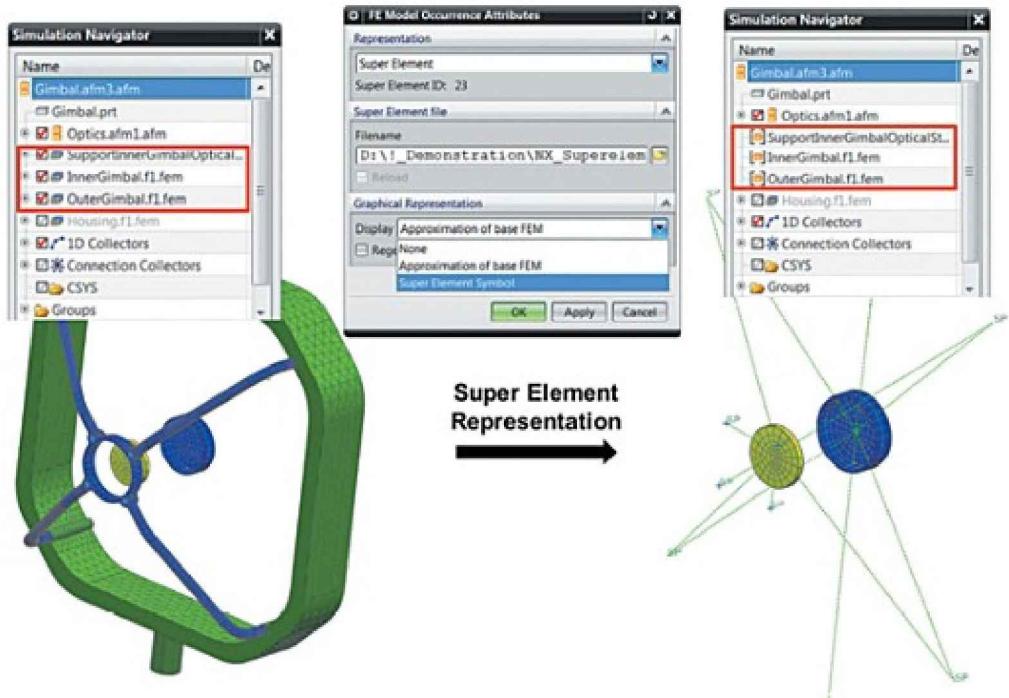


Figure 3.81. Using super elements in an assembly FEM

3.8. Example. Creating a finite-element model of a U-shaped frame

This example discusses the stages of creating a FE model of a U-shaped lifting gear frame (Figure 3.82). The example model consists of steel sheets (thin-shell part), as well as hydraulic cylinder fixtures and hinged bearing fixtures (thick-shell part). The FE model consists of hexahedral and tetrahedral elements describing the thick-shell part, shell-type finite elements describing the thin-shell part of the frame, and additional point mass elements that model the neglected elements of the structure.

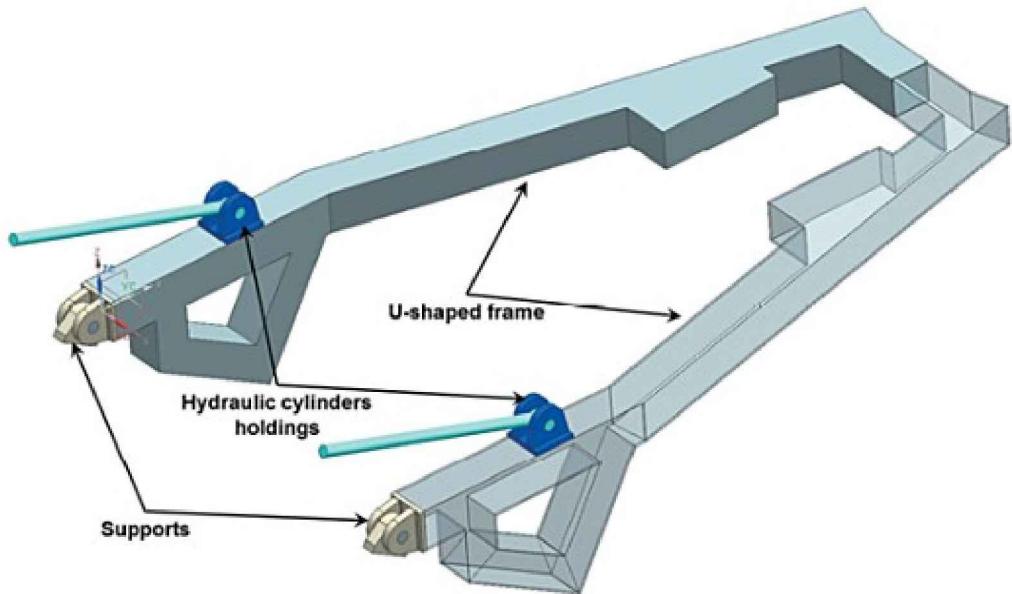


Figure 3.82. Geometry model of a U-shaped frame

The FE model is designed in several steps:

1. Importing the geometry model and creating a new FE model.
2. Creating midsurfaces.
3. Specifying FE mesh continuity conditions
4. Creating a 2D mesh. Specifying physical properties.
5. Creating a 3D mesh. Specifying physical properties.
6. Modeling neglected elements of the structure.
7. Setting up display.

In this example most commands are accessed using toolbars. The toolbars are arranged according to the meaning of the tools. To open the required toolbar (if it is not displayed), right-click in any location except the graphics window, and select the name of the toolbar in the list of available toolbars to display it. All toolbars necessary for using NX Advanced Simulation are listed in the lower part of the toolbar list.

3.8.1. Importing the geometry model and creating a new FE model

Run NX and create a new model file. To do this, choose main menu command: **File→New...**, and make sure you specify millimeters as the units of measurement of the new model (Figure 3.83). Enter *Frame.prt* as the new

file name and specify the folder to save it, then click **OK**.

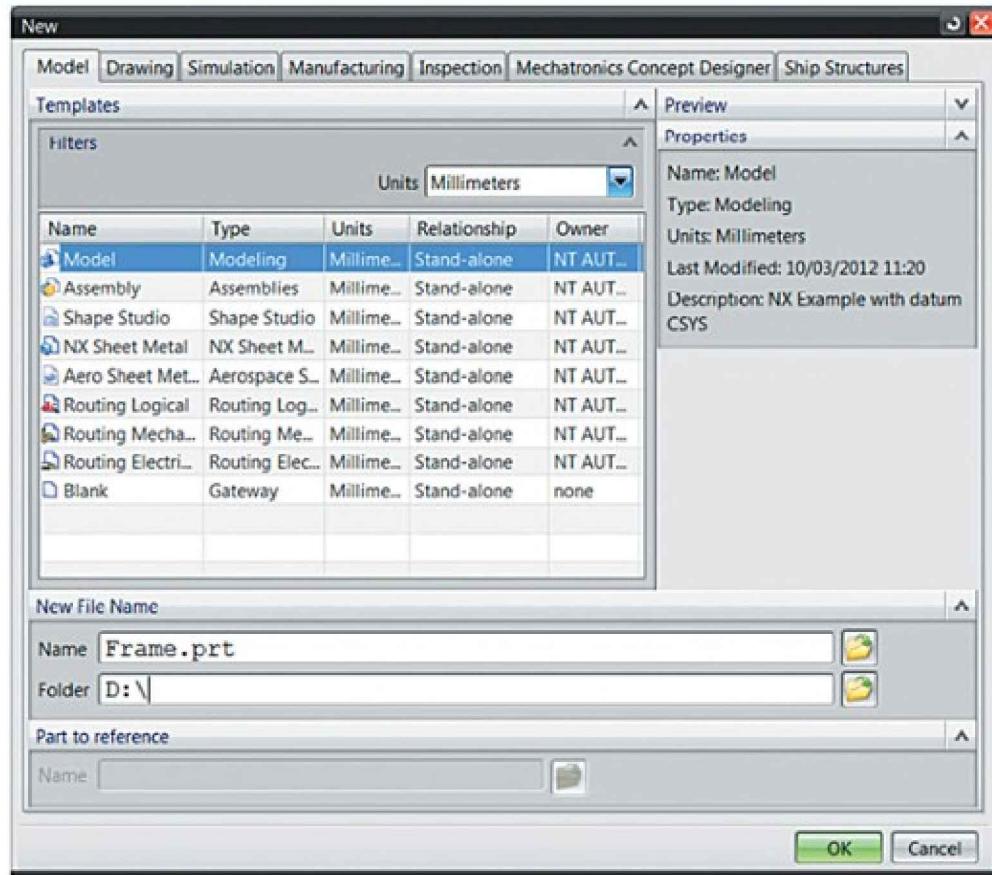


Figure 3.83. New part creation dialog box

To import a geometry model choose **File→Import→Parasolid...** in the main menu, select the *Frame.x_t* file, then click **OK**. Choose **File→Save**.

Go to NX Advanced Simulation module (**Start→Advanced Simulation**).

Set default options for all dialog boxes. To do this, choose **Preferences→User Interface→General→Reset Dialog Box Settings** in the main menu, then click **OK**.

Right-click the model in **Simulation Navigator** or **Simulation File View** dialog box. Select **New FEM** to create the new model. In the FEM file creation dialog box, make sure you select **NX Nastran** as the template, enter the name of the new FE model file (*Frame_fem1.fem*) and specify the folder

to save it in (Figure 3.84, A), then click **OK**. In the new FE model creation dialog box (Figure 3.84, B), make sure you select the *Create Idealized Part* dialog box and click **OK**.

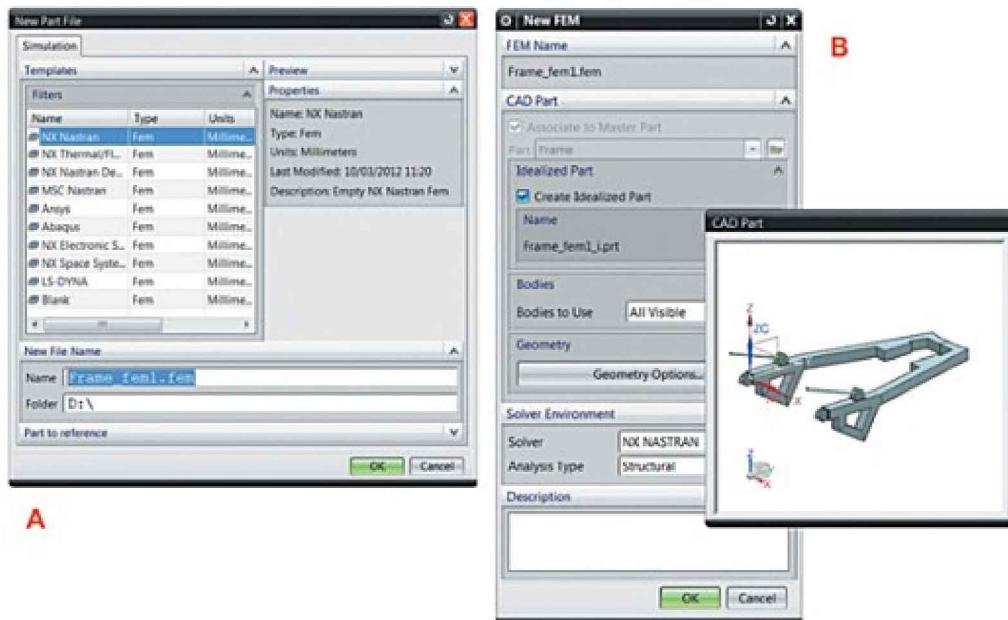


Figure 3.84. Creating a new FE model

You created an idealized model and an empty FE model with the corresponding files based on the source geometry model. In the **Simulation File View** window you can see the names of three model file names. The FE model file that is currently active is highlighted in blue.

3.8.2. Creating midsurfaces

The next step involves creating midsurfaces for the thin-shell part of the structure as part of the idealized model.

Double-click the *Frame_fem1_i* file in the **Simulation File View** dialog box to go to the idealized model. You get a warning to create an associative copy of the geometry model if you want to perform any geometry operations. Click **OK**. Click **Promote** on the **Advanced Simulation** toolbar to create an associative copy.

To create midsurfaces, use the **Midsurface** toolbar and use the **Midsurface by Face Pairs** command. In the dialog box, select two thin-shell bodies of

the U-frame (Figure 3.85), click *Automatically Create Face Pairs*, in the *Display Options* select the *Hide Solid Body Upon Apply* checkbox, and click **OK**.

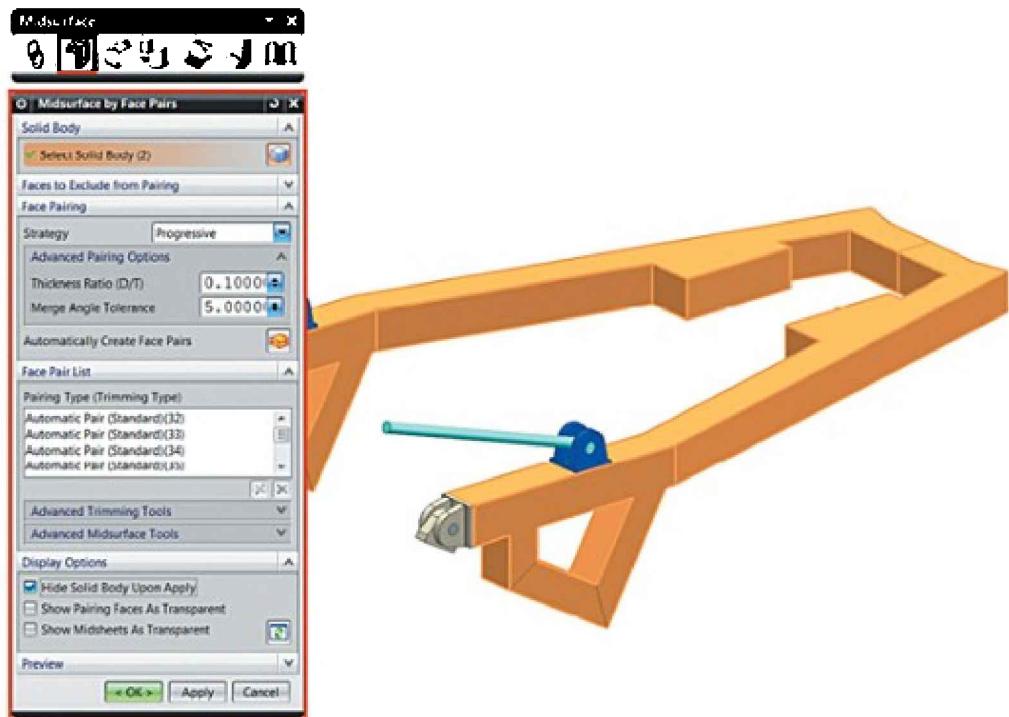


Figure 3.85. Creating midsurfaces

You created midsurfaces (sheet bodies) for the thin-shell bodies. The solid bodies that are not used in the subsequent stages are hidden.

3.8.3. Specifying FE mesh continuity conditions

Double-click the *Frame_fem1* file in the **Simulation File View** dialog box to go to the FE model. Hide the unused thin-shell solid bodies. To do this, expand the *Polygon Geometry* node in the model tree, find the relevant bodies (16 and 19) in the list and clear the check box.

Before you create the finite-element model, stitch the relevant midsurface to enforce continuity of the shell mesh, and specify mating conditions for 3D meshes.

Select the **Stitch Edge** command (**Model Cleanup** toolbar) to highlight the edges corresponding to free boundaries of sheet bodies in the graphics

area. Use automatic stitching (Figure 3.86) by selecting all polygon bodies of created midsurfaces and clicking **Apply** to make sure there are no free boundaries between the sheet bodies. Click **OK**.

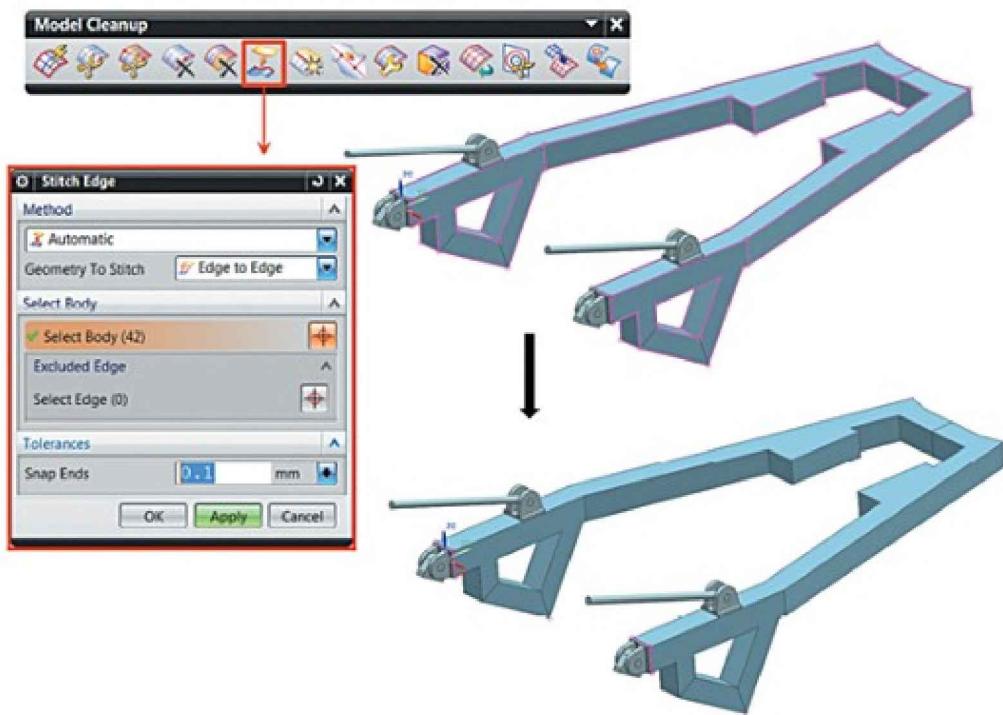


Figure 3.86. Stitching free boundaries

To create 3D mesh mating conditions, open the **Mesh Mating Condition** dialog box, use the *Automatic Creation* type to select all 3D polygon bodies except bushings (cylinder elements). Leave default values for all other parameters (Figure 3.87), and click **OK**.

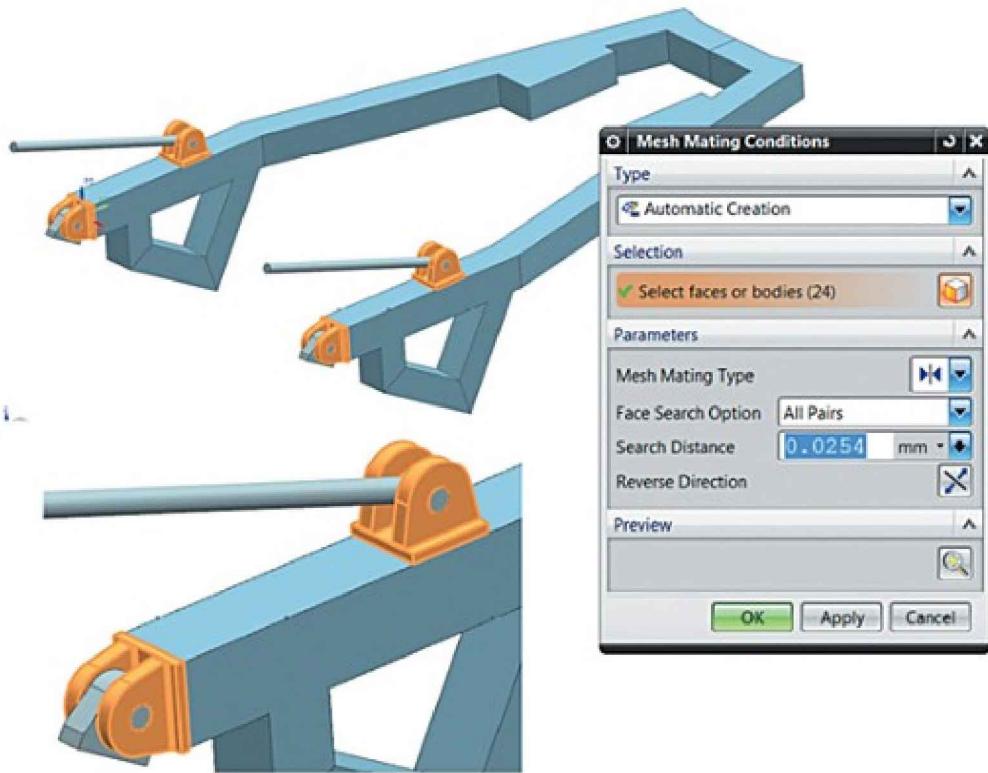


Figure 3.87. Creating mesh mating conditions

The model is prepared for FE mesh creation. Shell elements are used for sheet bodies, and hexahedral/tetrahedral elements are used for all other bodies.

3.8.4. Creating a 2D mesh. Specifying physical properties

Select the **2D Mesh** command (**Finite Element Model** toolbar) and set up the following options (Figure 3.88):

- Use *Select Objects* to select all polygon bodies of midsurfaces.
- Under *Type (Element Properties)*, select the **CQUAD4** element type.
- Select the *Split* mesh generation method to create uniform elements using a general element size.
- Under *Element Size* enter the general element size of **50 mm**.
- Select *Attempt Free Mapped Meshing* to create a regular mesh where possible.
- In the *Mesh Settings* group, make sure you select the *Export Mesh*

to Solver check box.

Click **OK**.

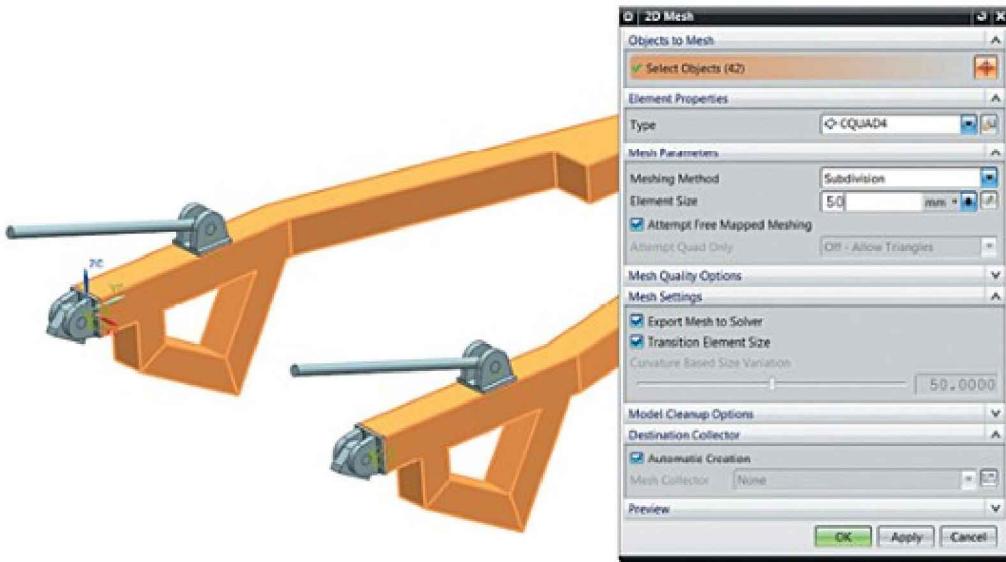


Figure 3.88. Creating a 2D FE mesh

Creating a 2D mesh is not enough, you also need to set up physical properties, including element thicknesses and material properties.

In the **Simulation Navigator** model tree right-click the 2D collector **ThinShell(1)** and click *Edit*. In the **Mesh Collector** dialog box, change collector name to **Frame** (Figure 3.89, A), then click **Modify Selected** (next to **Shell Property**). In the new **PSHELL** physical property editing dialog box (Figure 3.89, B), click **Choose Material** (next to **Material 1**). In the **Material List** dialog box, select **Steel** as the material. Click **OK** in all dialog boxes.

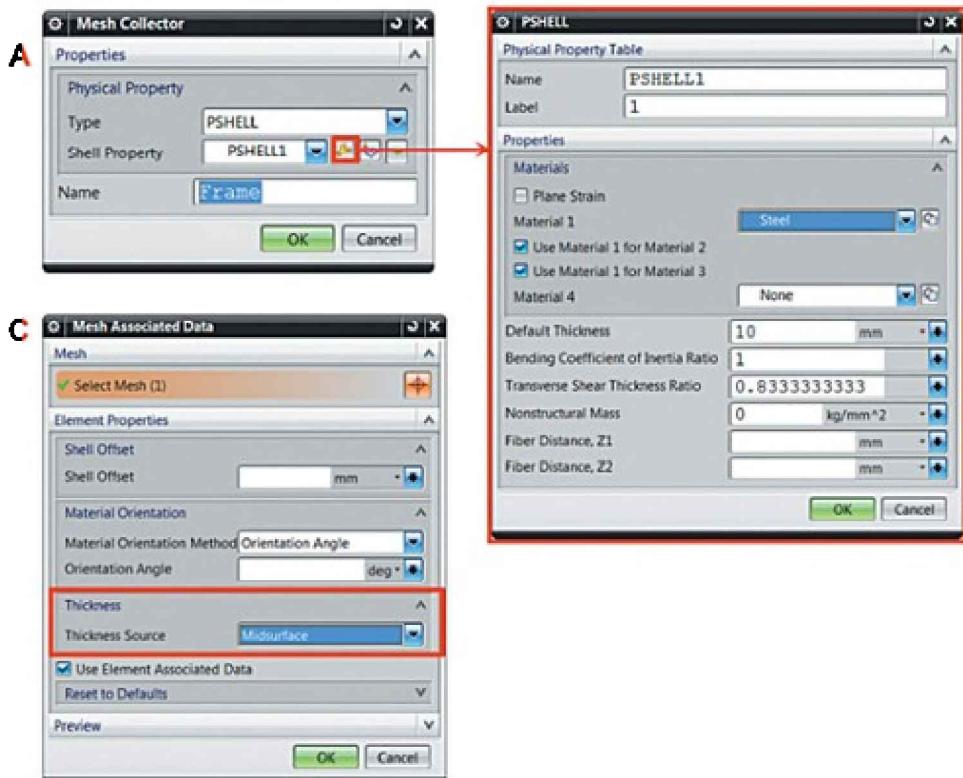


Figure 3.89. Specifying physical properties

Because shell elements were created for midsurfaces of thin-shell bodies you simply need to click *2d_mesh(1)* of the *ThinShell(1)* collector to specify their thicknesses, and choose the **Edit Mesh Associated Data** command in the shortcut menu. In the **Mesh Associated Data** dialog box, in the *Thickness Source* parameter drop-down list, select *Midsurface* (Figure 3.89, C). Click **OK**.

3.8.5. Creating a 3D mesh. Specifying physical properties

Now generate a 3D mesh for the rest of the structure. Select the **3D Swept Mesh** command (**Finite Element Model** toolbar) and set up the following options (Figure 3.90):

As the *Type*, select *Multi Body-Infer Target*.

Under *Select Source Face*, select one end face of each body except hydraulic cylinder elements.

Under *Type (Element Properties)*, select *CHEXA(8)*.

In the *Source Element Size* box, enter 20 mm.

Select the *Attempt Free Mapped Meshing* check box.

Select the *Edge Mapping* check box.

Click **OK**.

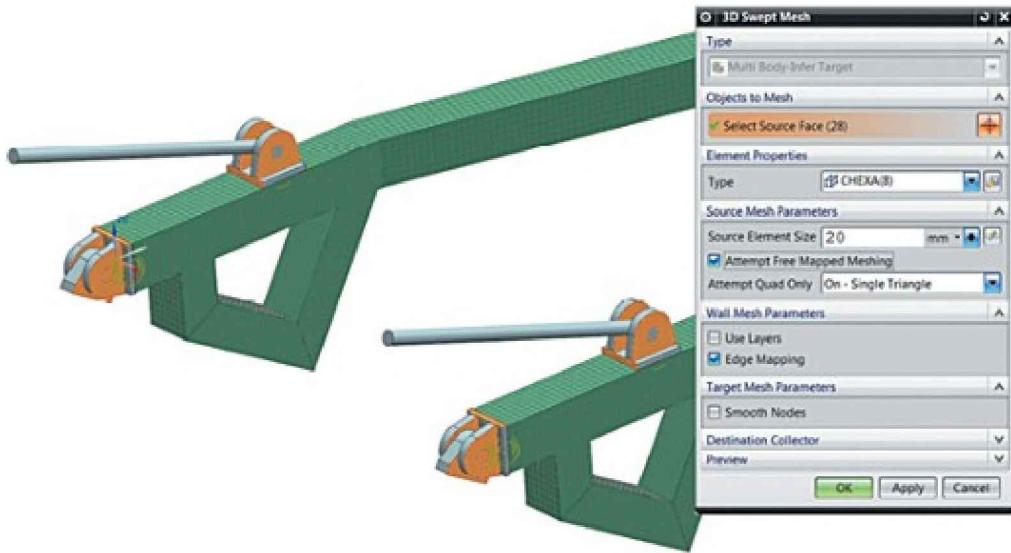


Figure 3.90. Creating a hexahedral mesh

To create the mesh on the rest of the structure, use the

3D Tetrahedral command (**Finite Element Model** toolbar), and set the following options (Figure 3.91):

- Use *Select Bodies* to specify two bodies that are not divided into elements.

Under *Type* (*Element Properties*), select *CTETRA(10)*.

In the *Element Size* box, enter 20 mm.

Automatic creation (Destination Collector): clear the checkbox and under *Mesh Collector*, select the created *Solid(1)*.

Click **OK**.

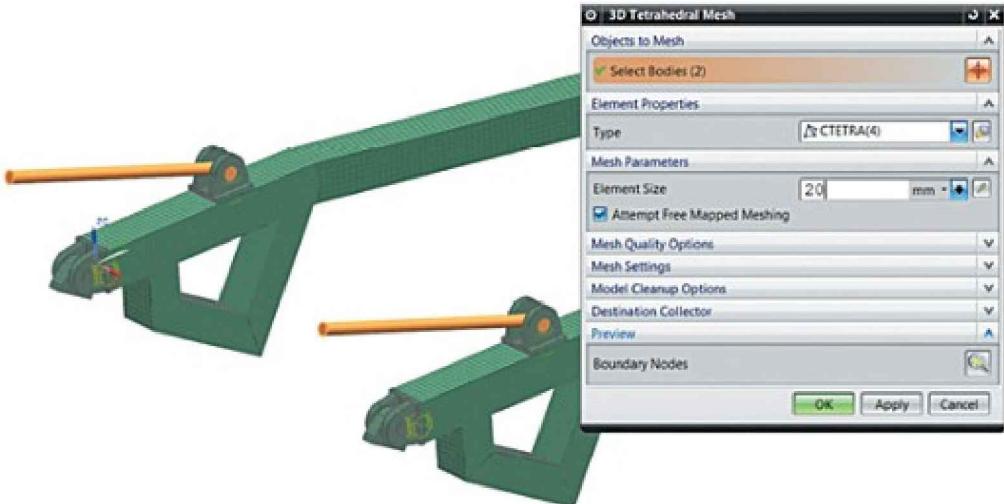


Figure 3.91. Creating a tetrahedral mesh

Because the material for shell elements is already selected, just specify it for the new 3D mesh. To do so, right-click the *Solid(1)* collector in the model tree and choose *Edit*. In the **Mesh Collector** dialog box, change the collector name to *Solid*, and then click *Modify Selected* (next to *Solid Property*). In the new **PSOLID** physical property editing window, in the *Material* drop-down list, select *Steel*.

3.8.6. Modelling neglected elements of the structure

Additional parts of the structure that are not considered in the simulation model can affect its stress-strain state. These parts can be replaced with point masses to retain all inertial properties of the structure. Create point mass elements in areas where neglected parts of the structure attach, and connect these elements to attachment areas using rigid link elements.

Hide the 2D finite-element mesh and use the *Static Wireframe* view to simplify subsequent manipulation of the FE model.

Use the **1D Connection** command from the **Advanced Simulation** toolbar. In the dialog box, specify the following parameters (Figure 3.92):

Under *Type*, select *Point to Face* to create a set of RBE2 elements from a point to all nodes of the specified face.

Under *Source*, enter the **Point Construction** menu and create a point using

the *Between Two Points* type. Select two opposite points on the face (Figure 3.92, A). Click **OK**;

Under *Target*, select a face as shown in Figure 3.92, B.

In the *Connection Element* group, in the *Type* drop-down list, select *RBE2*. Click **Apply**.

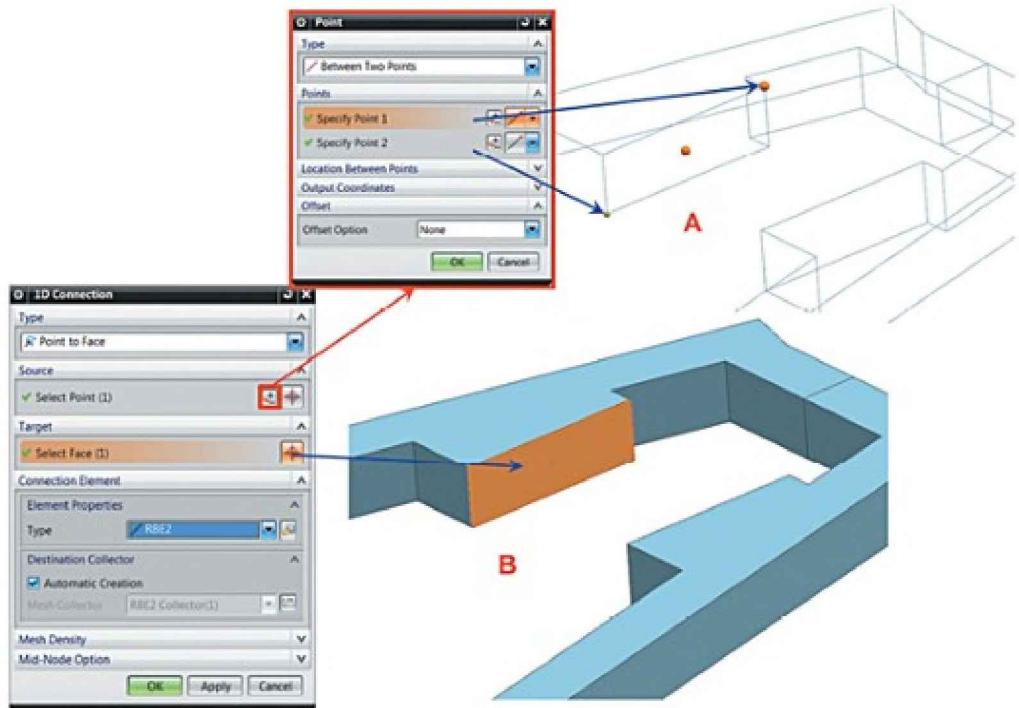


Figure 3.92. Creating 1D connections with face nodes

Repeat the 1D connection creation process for the opposite symmetrical face.

Create a 1D connection through the centre point and 4 edges as shown in Figure 3.93, using the **1D Connection** command with the *Point to Edge* type.

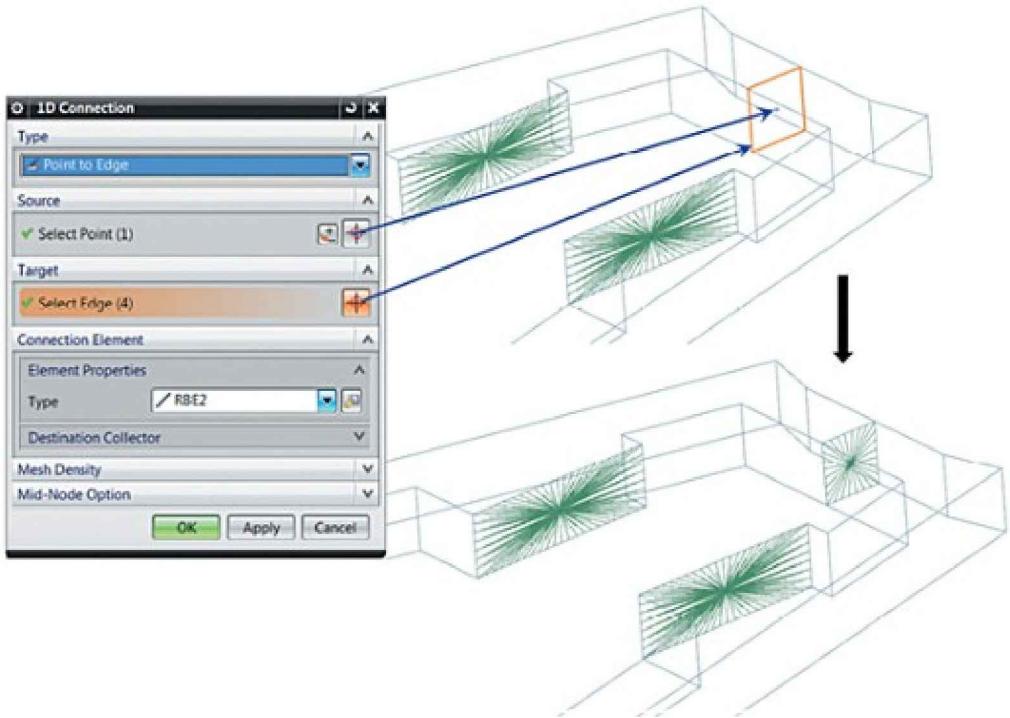


Figure 3.93. Creating 1D connections with edge nodes

Create two point mass elements with 800 kg mass and one point mass element with 500 kg mass. Use the **0D Mesh** command (on the **Advanced Simulation toolbar**) to specify two points created at the previous step that are connected to nodes on faces (Figure 3.94, A). Select CONM2 element type. Do not leave the dialog box. Select the **Mesh Associated Data** command and enter 800 kg as the *Mass* value. Click **OK**, click **Apply**. Repeat the creation process for the 500 kg point mass element in the previously created point that is connected to edge nodes (Figure 3.94, B). Click **OK**.

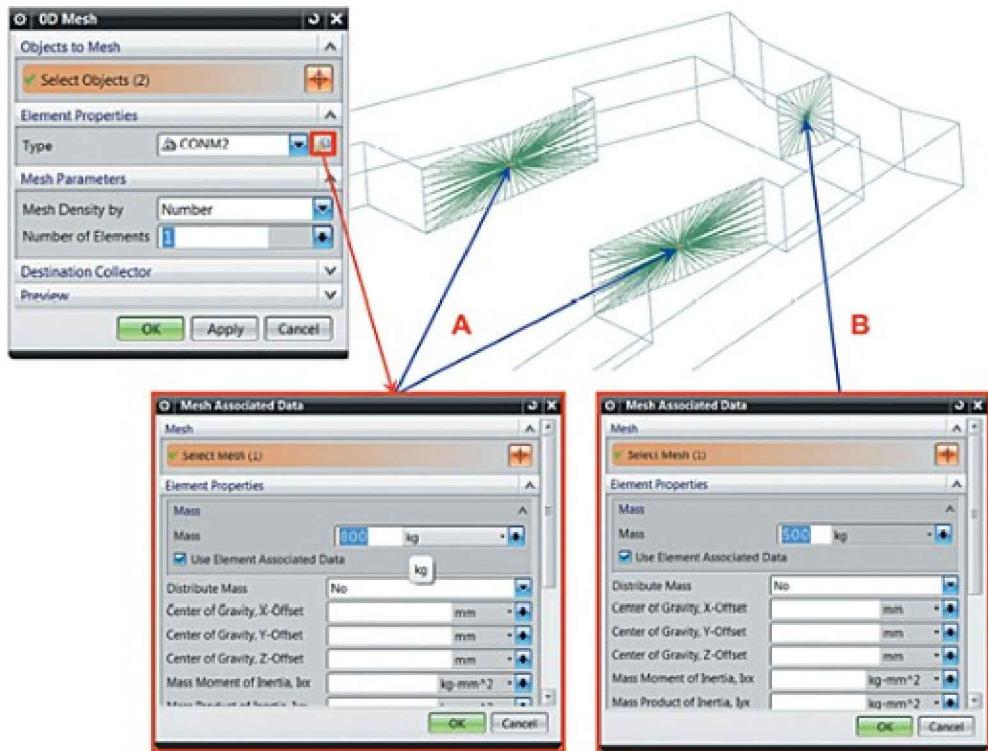


Figure 3.94. Creating point masses

Note that the FE model you created is not connected in the area where 3D elements are mated with 2D elements. At the simulation model creation stage, in addition to specifying boundary conditions, you also need to specify conditions for gluing 3D elements with faces of 2D elements to model the structural elements that attach the frame to hydraulic cylinders. You also define conditions for gluing 3D element faces with 2D element edges in the area where the frame is attached to hinge bearings.

3.8.7. Setting up display

Display all FE meshes and select **Shaded with Edges**. Shell elements are displayed with zero thickness by default. To check whether thickness is specified correctly, use the **Edit Display** command. To use this command, right-click the mesh or the 2D mesh collector by the name of *Frame* in the model tree in **Simulation Navigator**. In the dialog box (Figure 3.95, A), set the *Element Thickness and Offset* check box and click **OK**.

Display the FE model with different colours for different meshes. To do this, in the main menu choose **Preferences→Model Display**, in the dialog box

(Figure 3.95, B), in the *Element* tab, under *Color Basis*, select *Mesh*. Click **OK**.

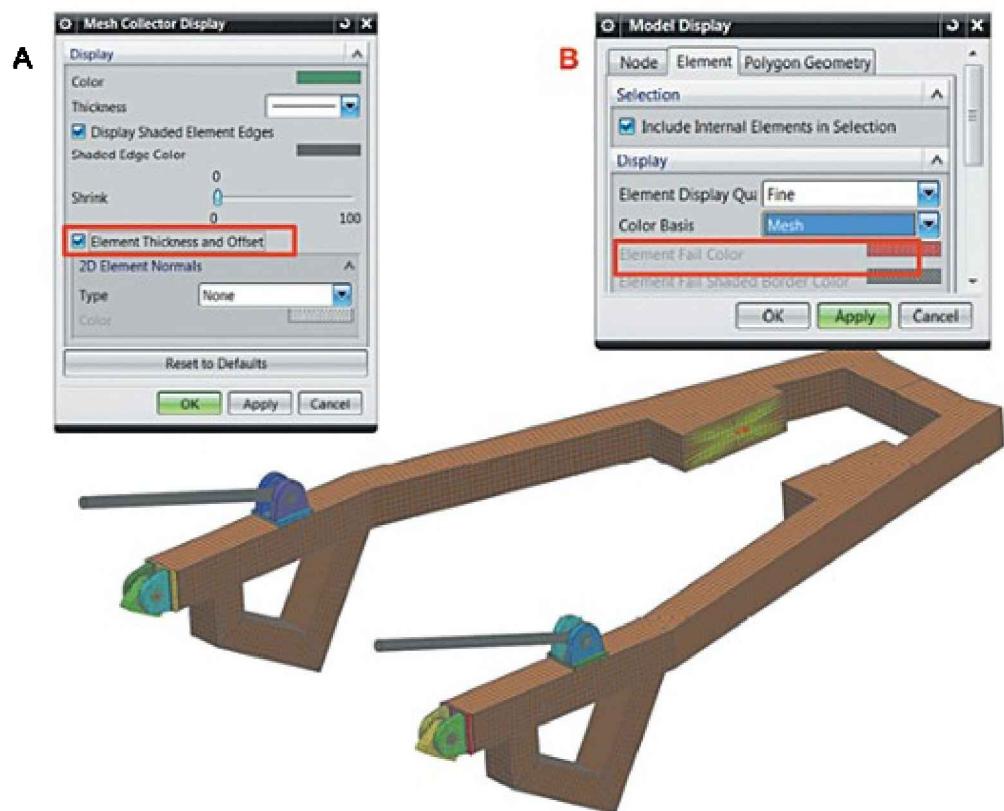


Figure 3.95. FE model display settings

Save the model. To do this, in **Simulation Navigator**, right-click **Frame_fem1** in the **Simulation File View dialog box** and choose **Save**.

Chapter 4. Creating a simulation model

The last step before simulating the structure involves working with the simulation model. This model corresponds to the **.sim* file that you created based on the FE model. This simulation file contains all behavior parameters and properties of the structure, simulation cases, solver options such as solution type, solution step, simulation objects (contact boundary conditions and so on), loads, constraints, and physical properties that can be overwritten in the *.sim* file. You can create several simulation files for each FE model file.

To access the simulation model preparation commands, use the **Simulation Navigator** tab of the Resource Bar, the **Advanced Simulation** toolbar or choose the relevant commands in the main menu. In this chapter, you will use only those commands that are relevant to deformable solid mechanics problems.

Figure 4.1 shows the **Advanced Simulation** toolbar. When a simulation file is active, the toolbar contains commands for working with loads, constraints, modeling and simulation objects, as well as commands for running the simulation and creating reports. Although the **Physical Properties**, **Mesh Associated Data**, and **Finite Element Model Check** commands are rather applicable to the finite-element model (**.fem* file), you can also use these with the simulation model. In the latter case, however, functionality is limited.

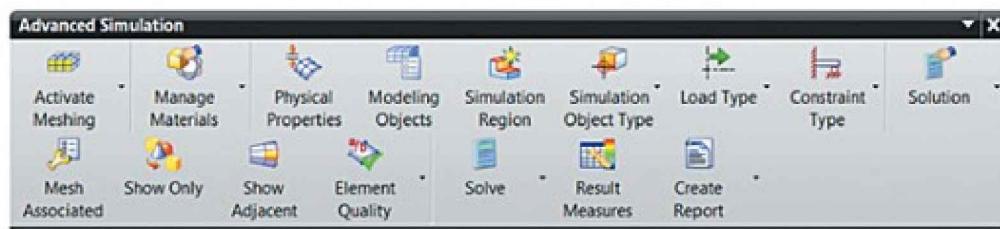


Figure 4.1. Commands of the Advanced Simulation toolbar, simulation file

4.1. Structure of the simulation model. Simulation Navigator

You can access the simulation model in one of the several ways. The fastest is using the **Simulation File View** window on the **Simulation Navigator** toolbar where all four files of the model are generally displayed: the master

model, the idealized model, the finite-element model, and the simulation model. Double-click the simulation model to activate it. The corresponding file is highlighted with blue. Note that if you use a new simulation file, the model tree in the **Simulation Navigator** toolbar is populated with empty containers (nodes) that are filled when the full simulation model is created.

Before studying methods and special aspects of working with simulation models in **NX Advanced Simulation**, you need to become familiar with the **Simulation Navigator** tab of the Resource Bar. This tab represents the current model in a tree view and contains links to all objects that comprise the model. You can use this tab to manage key data of the simulation model:

- View the structure and content of the model.
- Quickly switch to the FE model, idealized model, or master model.
- Create new objects, such as new boundary conditions, data fields, and coordinate systems.
- Manage the display of certain objects in the graphics area.
- Edit, rename, and delete existing objects.

The tree view contains two logical parts: the first part is responsible for all specified objects (Figure 4.2, A), while the second part contains objects that are used in a particular solution (Figure 4.2, B). Here is a brief description of each node of the model tree:

- *Name_fem.fem* is the link to the FE model, including the following FE objects: polygon geometry, parameters, and mesh collectors.
- *CSYS* is the list of local coordinate systems created in the simulation file and the FE model.
- *Groups* is the list of groups created in the simulation files and the FE model.
- *DOFSets* is the list of degrees of freedom in particular nodes that can be used, for example, when working with super elements.
- *Regions* is the list of areas or regions for specifying contact interactions or gluing connections.
- *Fields* is the list of data fields created to specify propagation of values.
- *Simulation Object Container* is the list of all simulation objects, such as contact pairs, initial conditions, and so on.
- *Load Container* is the list of all specified loads.

- *Constraint Container* is the list of all specified constraints, that is, conditions set on degrees of freedom.
- *Solution 1* (for example, *Static*) is a solution created for performing an analysis of some kind. There can exist an arbitrarily large number of solutions, and they can correspond to a single type of analysis, or to many different types. Each solution contains simulation model objects that are unique to it. These objects are also present in the main part of the model tree.

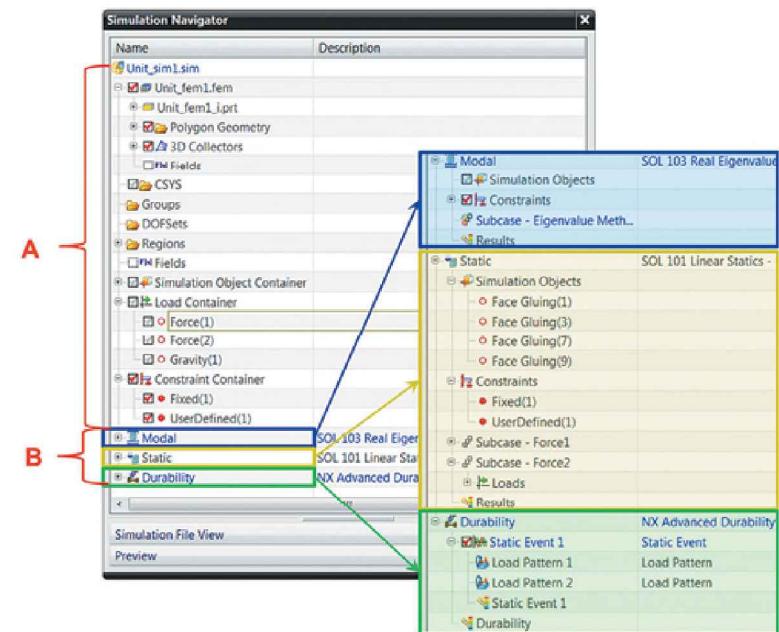


Figure 4.2. Simulation Navigator tab

You can use the **Simulation Navigator** of the active simulation model to access all commands you need for preparing the calculations. To call a command, right-click the relevant node and choose the command in the shortcut menu (Figure 4.3).

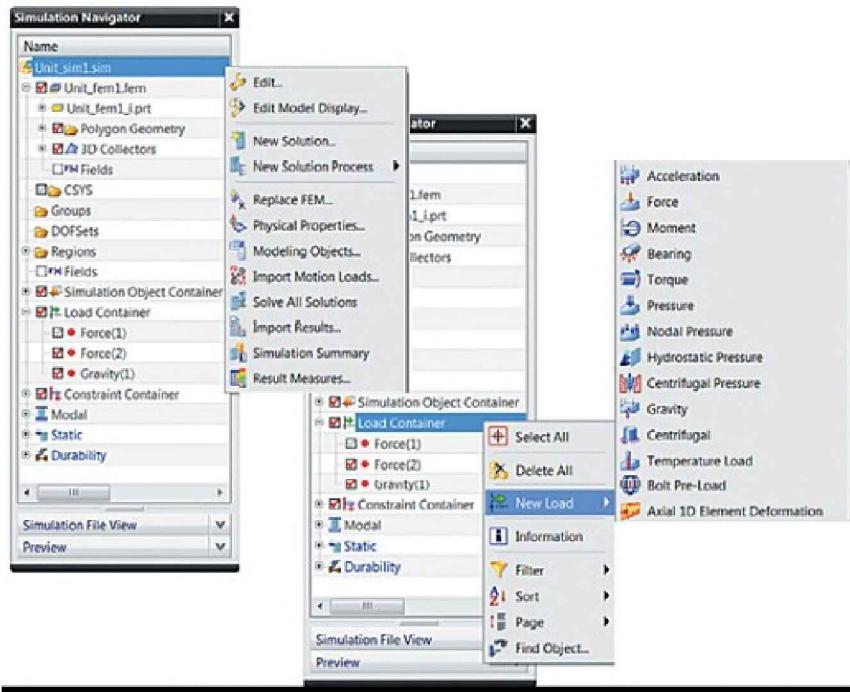


Figure 4.3. Commands on the Simulation Navigator tab

The simulation model contains many objects, complicating navigation of the graphics area. To make selecting objects for commands easier, you can filter the objects by type or use the show/hide capabilities of the model tree view. To manage the display of various types of objects in the graphics area in **Simulation Navigator**, click the red indicator (check mark) to temporarily hide the object. To show the object, click the gray indicator.

In addition to creating new objects, you can use the **Simulation Navigator** to edit, rename or delete existing objects, or to edit their display options. The **Simulation Navigator** also supports using the so-called drag-and-drop interaction to quickly reorganize the objects in the model:

- You can drag meshes from one collector to another.
- You can add simulation objects, loads, and boundary conditions to solutions.

To eliminate any errors related to particular solution conditions, there are indicators for simulation objects, loads, and constraints in the **Simulation Navigator** tab that show their current status (Figure 4.4):

- A filled red circle indicates a fully defined object that participates in the current solution.
- An unfilled red circle also indicates a fully defined object that does not, however, participate in the current solution. To add it to the solution, drag it or right-click the relevant object and choose the *Add to active solution* or *step* command in the shortcut menu.
- A crossed-out circle indicates an object that is not fully defined and is therefore ignored in the solution. This can occur when geometry is modified or meshes are rebuilt, possibly causing a loss of polygon geometry or nodes/elements that were used as the basis of simulation model objects. If this is indeed the case, use the underdefined object editing mode to set new parameters.

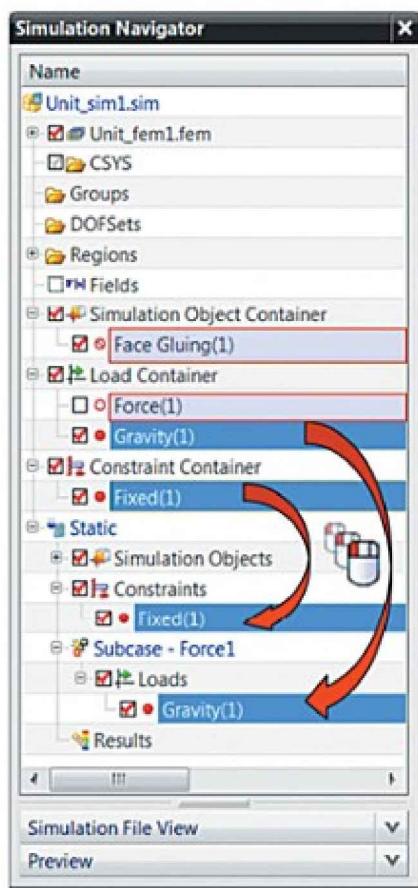


Figure 4.4. Status of simulation model objects

4.2. Coordinate systems and data fields

To fully understand the methods of setting up loads and constraints, you need to consider the existing coordinate systems and look at data field creation methods that are used in creating simulation models.

4.2.1. Coordinate systems

Coordinate systems are not only an inseparable and helpful component of geometry model creation process., but also the primary tool for creating simulation models and processing analysis results.

Five types of coordinate systems are used in the simulation model when working with loads and constraints (Figure 4.5):

1. The global coordinate system is automatically set up by the software, it is always Cartesian and immutable. It is also known in NX as the absolute coordinate system. All other coordinate systems are defined relative to this one. When you export the FE model into an input file of the solver, all node locations are recorded with global coordinates.
2. The work coordinate system (WCS) is used as a mobile coordinate system as opposed to the global system. It is an auxiliary system for most operations on the geometry model as well as the simulation model. This system can be arbitrarily positioned and oriented in space. It is always Cartesian and coincides with the global coordinate system by default.
3. Local coordinate systems are defined by the user and allow working with geometry and finite-element operations. When you create a local coordinate system, you need to select one of the types: Cartesian, cylindrical, or spherical.
4. Nodal coordinate systems are local coordinate systems assigned to a single node or a set of finite element nodes. By default, these coincide with the global coordinate system. There are two types of nodal coordinate systems:
 - The node displacement coordinate system, which defines nodal degrees of freedom.
 - The reference coordinate system, which serves as the reference for specifying or modifying node locations. For example, it is used when adding an FE model to a different existing model to quickly move nodes and elements by specifying a new location and orienting the reference coordinate system assigned to the nodes in question.

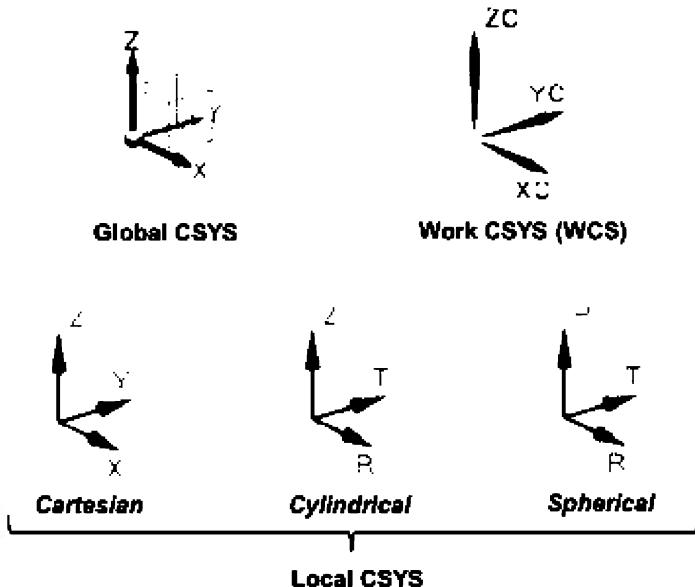


Figure 4.5. Coordinate systems

With regard to the simulation model, coordinate systems are necessary to specify component loads and set up degree of freedom constraints. For these purposes you can use local and nodal coordinate systems that you can create directly when using object definition commands or in advance, using **CSYS** commands for local coordinate systems and **Assign Nodal Coordinate System** for nodal coordinate systems. All local coordinate systems used in the simulation model are saved in the coordinate system node of the model tree in the **Simulation Navigator tab**. Right-click an existing coordinate system to choose a command that allows to modify the system, delete it, or set up display.

4.2.2. Data fields

In NX you can specify functions (data fields) that depend on one or more independent variables. In **Advanced Simulation** you can use functions to:

- Describe boundary conditions that vary with time, temperature, or frequency.
- Describe boundary conditions that vary in space. For example, a pressure value depends on where it is applied to the surface (Figure 4.6).
- Set up nonlinear properties of materials. For example, you can

define a stress-strain deformation curve for elastic-plastic and hyperelastic material models, and specify temperature-dependent properties.

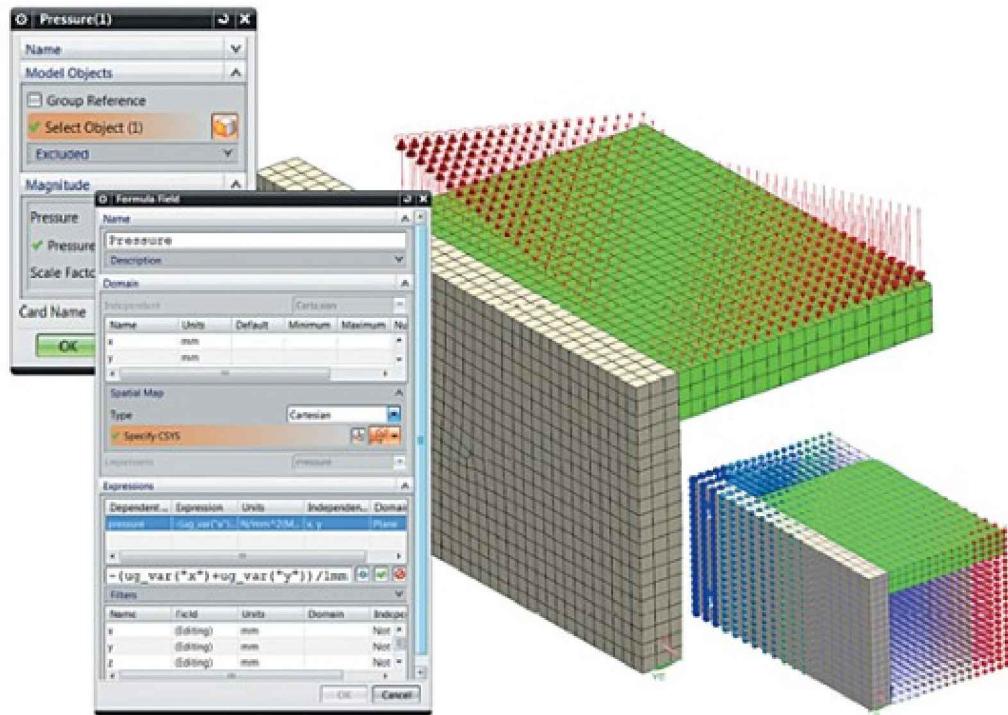


Figure 4.6. Spatial data field example

The ability to define dependencies as data fields for boundary conditions is important and relevant in many types of analyses. Therefore, before you study the process of defining boundary conditions, you need to become familiar with data field fundamentals.

Typically, data fields for values are defined directly when the value is defined. You can, however, call the relevant command through the **Fields** node of the **Simulation Navigator** or in the main menu, create the necessary function with the selected independent variables, and use it when defining boundary conditions. This method affords great flexibility in selecting independent and dependent variables but the function you create must be compatible with the subsequent command that you use to set boundary conditions or material properties. For example, when defining a force, you can use only frequency, temperature, and time as independent variables.

Data fields are created using either a formula (Figure 4.7, A) or a table of values (Figure 4.7, B). You can use data fields to define relationships between independent and dependent variables. You can also use other data fields as independent variables. All data fields are saved in the simulation model, and are added to the model tree. You can edit, rename, export, delete the fields or display them in the graphics area (Figure 4.8).

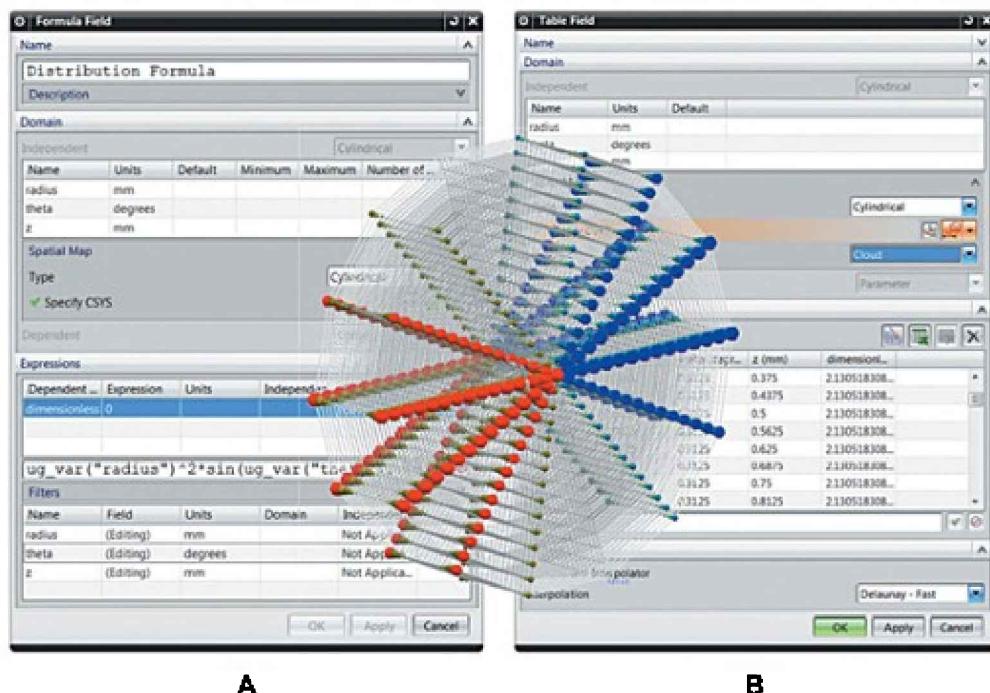


Figure 4.7. Creating data fields

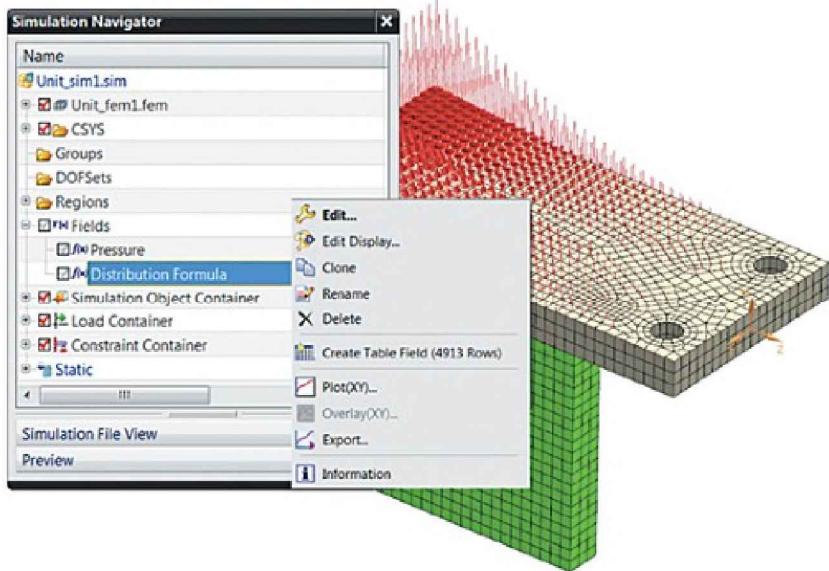


Figure 4.8. Data fields in the model tree

The following is a description of the most important options of the **Formula Field** dialog box that is used to create formula-type data fields (Table 4.1) and the **Table Field** dialog box that is used to create table-type data fields (Table 4.2). These dialog boxes have many common options, therefore Table 4.2 describes only relevant options.

Table 4.1. Options of the **Formula Field dialog box**

Domain	
Independent tab	<p>Domain Select the domain of variation of an independent variable</p> <ul style="list-style-type: none"> – Select independent variables: <i>Time</i>, <i>Frequency</i>, or <i>Temperature</i>. – Select independent variables in space: <i>Angle</i>, <i>Angle–Angle</i>, <i>Cartesian</i>, <i>Cylindrical</i>, <i>Length</i>, <i>Parameter</i>, <i>Parameter Plane</i>, <i>Parameter Space</i>, <i>Plane</i>, <i>Pole</i>, <i>Spherical</i>. – Select available variables for general domain type (1D, 2D, 3D), such as stress, acceleration, energy, and so on. In the general case you can use all physical values available in NX.
Dependent tab	<p>Domain Select domain of variation of the dependent variable. In essence, this is the function as such.</p> <p>This option is automatically determined when you set up the data</p>

	field for boundary conditions.
Variables	List of variables that can be used in the function.
Units	Specify units of measurement for the variable.
Borders Bounds	<p>Specify the variation range of function variables.</p> <p><i>Minimum</i> Specify the minimum value of the variable.</p> <p><i>Maximum</i> Specify the maximum value of the variable.</p> <p><i>Default</i> <i>Number of Points</i> Specify the default number of points for function output, such as plotting or saving in a solver file.</p>
Inclusive	If you select <i>Inclusive</i> , exact <i>Minimum</i> and <i>Maximum</i> values are used, otherwise the <i>Minimum</i> value is incremented by a system precision value (1e-10) and subtracted from the <i>Maximum</i> value. You can use this option to specify a range for points where the function may be undefined, for example for $\log(x)$ x the range can be defined as 0 (exclusive) to 1 inclusive.
Spatial Map	<p>Enabled for the region of variation of parameters in space.</p> <p><i>Type</i> Specify the method of distribution in the model space:</p> <ul style="list-style-type: none"> – Select <i>Global</i> to use the global coordinate system. – Select <i>Cartesian</i> to use Cartesian coordinate system. – Select <i>Cylindrical</i> to use cylindrical coordinate system. – Select <i>Spherical</i> to use spherical coordinate system. – Select <i>Parametric Space</i> to use the UVW parametric coordinate space. – Select <i>Parametric Plane</i> to use any surface based on UV parametric coordinates that vary from 0.0 through 1.1. – Select <i>Parametric Line</i> to use any curve based on the U parametric coordinate that varies from 0 through 1. <p><i>Specify CSYS</i> Specify the coordinate system to use for the domain of variation of spatial parameters.</p>
Expressions	
Listing expressions	Table of variables, units of measurement, and expressions that are used to define relations between independent variables.
Expression input string	Expression input. In NX each variable has its own predefined symbol, therefore, you can double-click the variable in the filter

	list to use it.
Accept Edit	Confirm the last entered expression and add it to list.
Cancel Edit	Discard the entered expression
Filters	
Filters list	List of values that can be used in the expression you enter.

Table 4.2. Options of the *Table Field* dialog box

Domain	
Tab Independent	See Table 4.1.
Dependent tab	See Table 4.1.
Variables	See Table 4.1.
Units	See Table 4.1.
Bounds	See Table 4.1.
Spatial Map	See Table 4.1.
Data Points	
	<p>Select the interpolation algorithm that is used to create a smooth function from imported data:</p> <ul style="list-style-type: none"> – Select <i>Linear</i> for a 1D domain. Determine the weighted value from two adjacent points. – Select <i>Nearest Neighbor</i> for 1D, 2D, and 3D domains. Set the value by closest point. – Select <i>Inverse Distance Weighting</i> for 1D, 2D, and 3D domains. – Select <i>Delauney</i> for 2D and 3D domains. – Select <i>Renka's Modified Shepard</i> for 2D and 3D domains. <p>For details, see the NX documentation.</p>
Import from file	Import tabular values from a text file or a CSV (comma-separated values) file.
Edit table in spreadsheet	Edit table values in Excel
Show Table Rows	Hide table display. You can use this option when loading a large data array to save time.
Point input string	Enter comma-separated tabular data for columns in one string. To enter several strings, use semicolon as the separator.

Accept Edit	Confirm the values entered into the table.
Cancel Edit	Delete the entered values.

4.3. Loads, constraints on degrees of freedom, and simulation objects

The stage that involves applying loads and boundary conditions is critical because the simulation result is a solution of the equation system that simulates the structural behavior in operating conditions, that is, under the specified loads and boundary conditions.

All parameters and options of boundary conditions are interactive and are activated in accordance with the selected solver and solution type. You can create boundary conditions both before and after creating a solution. If the solution is created first, the loads, boundary conditions, and simulation objects are saved in the corresponding containers of **Simulation Navigator**: *Load Container*, *Constraint Container*, and *Simulation Object Container*. The objects you create are also saved in the corresponding containers of the current solution. If loads, constraints, and so on are created first, they are stored only in **Simulation Navigator** containers. Afterwards, you can drag them into created solutions and simulation cases (Figure 4.4).

You can apply boundary conditions to geometry objects (edges, faces, vertices, points) and FE model objects (nodes, elements, element faces and edges). In particular, boundary conditions applied to FE model elements are indispensable for working with imported meshes without parent geometry. You can create boundary condition values using constants, NX expressions, or data fields. Data fields can describe the way the values vary with time, temperature, frequency, or spatial dimensions (para. 4.2).

This chapter only describes the types of loads/boundary conditions and methods of applying these in the context of deformable solid body problems for NX Nastran solver. You can find a detailed description of other classes of problems in the second part of this book.

Loads and boundary conditions can be classified as either force conditions (forces and moments) or kinematic conditions (constraints on degrees of freedom). You can call the command that applies a particular load (Figure 4.9) using:

- The Advanced Simulation toolbar.
- The model tree view of the **Simulation Navigator** by right-clicking the *Load Container* node or the *Constraint Container* node and choosing *New Constraint* or *New Load* command respectively in the shortcut menu.

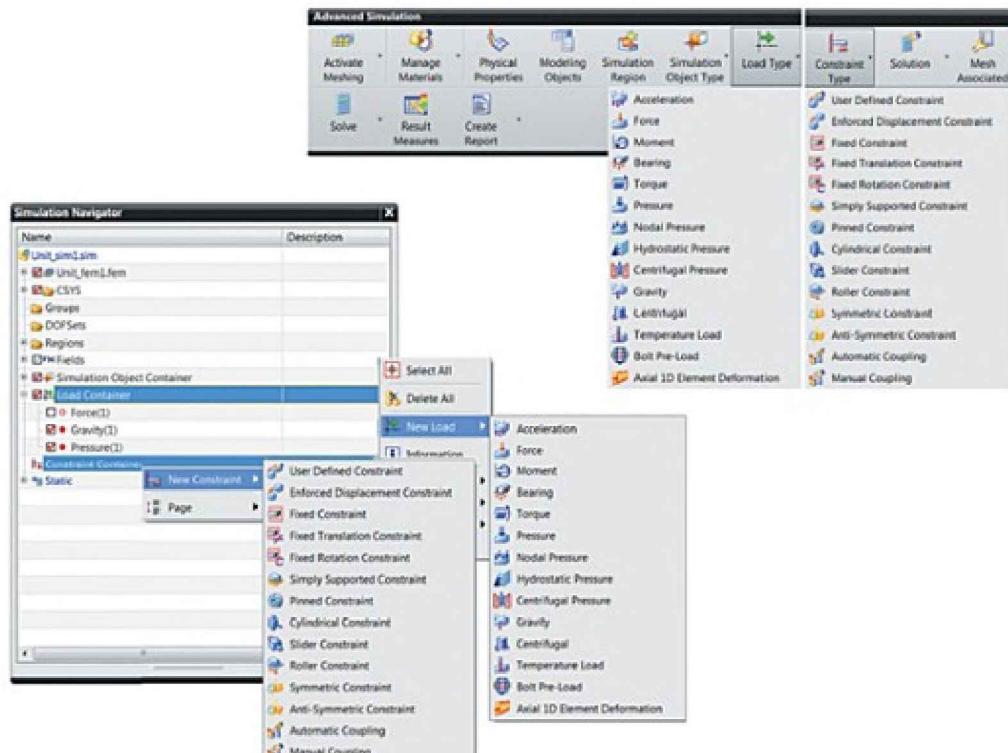


Figure 4.9. Using commands that apply loads and constraints

The structure of dialog boxes used for applying loads and constraints is the same. To start, select the type of the task, then select objects to which you need to apply the condition. Then, depending on the selected type, you can specify a coordinate system and corresponding component values or resultant values as well as the load direction. The magnitude of loads or constraints can be constant (*Expression*), or it can vary with time, frequency, or temperature. In the latter case you need to select *Field* and specify a data field. You can manage load and constraint application by selecting a distribution method:

- Select *Total Per Object* to apply the magnitude value to each selected object. For example, the resultant force value of 1000 N

is applied to three selected faces. This means each face is loaded with 1000 N regardless of its area.

- Select *Geometric Distribution* to evenly distribute the magnitude value to selected objects depending on their surface area. For example, the resultant force value of 1000 N is applied to two selected faces with 40 mm^2 and 60 mm^2 surface areas. This means the first face is loaded with 400 N, and the second face is loaded with 600 N.
- Select *Spatial* to distribute the load in space with a specified data field.
- When you specify a value, pay attention to the length unit that is selected from the drop-down list you find right of the value box. If you use data fields to define loads, constraints, and material properties, select one of the four setup methods next to the *Specify Field* option:
 - Choose *Select Existing Field from List* to specify an existing data field (Figure 4.10, A).
 - Select *Formula Constructor* to create a new data field and specify a formula that describes the relation between the dependent variable and independent variables (Figure 4.10, B).
 - Select *Table Constructor* to create a new data field by specifying tabular values and selecting an interpolation method (Figure 4.10, C).
 - Select *Link Constructor* to specify an existing data field as the reference field with modified spatial distribution method (Figure 4.10, D).

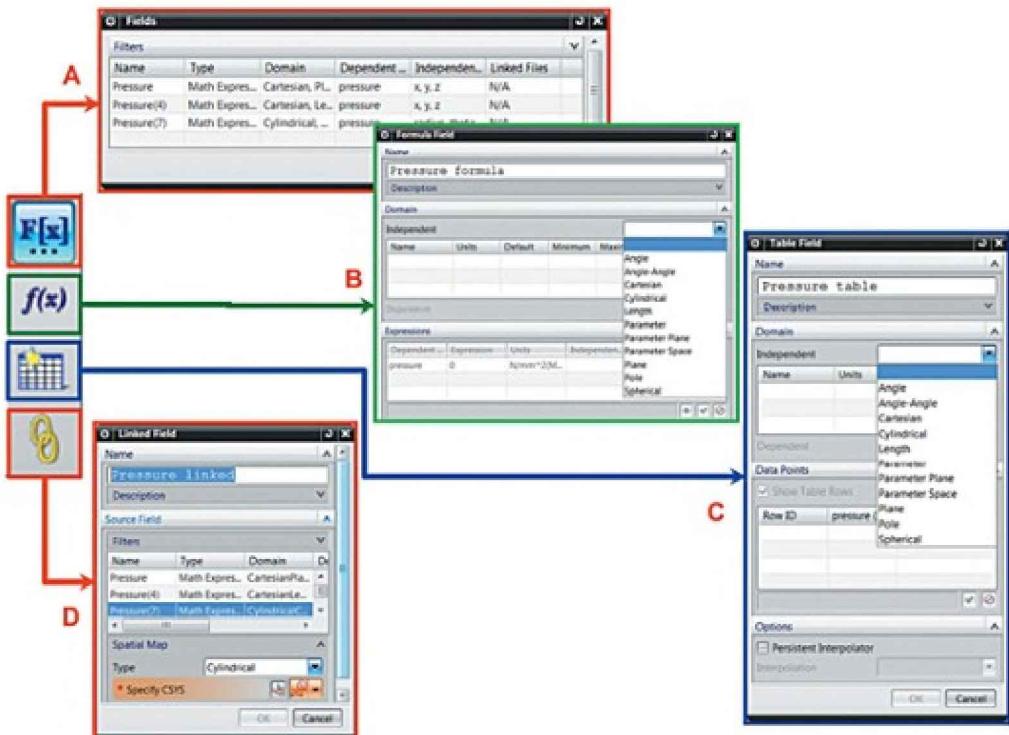


Figure 4.10. Four ways to specify data fields

4.3.1. Applying loads

In this section we will discuss the types of loads available for solving deformable solid body mechanics problems (Figure 4.11) in NX Nastran. Note that when you select objects for a single command, they must be of the same type. For example, if you select some nodes, you cannot additionally select faces.



Figure 4.11. Load types

You can use the **Force and Moment** commands to apply forces and moments. The options of dialog boxes are the same:

- Select one of the following *Type* options for specifying values:
 - Select *Magnitude and direction* to specify the resultant value and direction of force (Figure 4.12, A).
 - Select *Normal* to apply the force along the normal vector to the selected faces with a specified resultant value.
 - Select *Components* to specify force component values for the given coordinate system (Figure 4.12, B).
 - Select *Node ID Table* to use a data field containing node numbers and scale factors for the specified value for each of the three components of the given coordinate system.
 - Select *Edge–Face* to apply the force to a selected edge. You can use an auxiliary face to specify in-plane and out-of-plane components of this face.
- Specify objects, values, and directions in accordance with the selected force application type.
- Select *Distribution* method to use for objects (see above).

You can use this command to apply any kind of force. With the other commands you can specify particular types of forces in a simplified way.

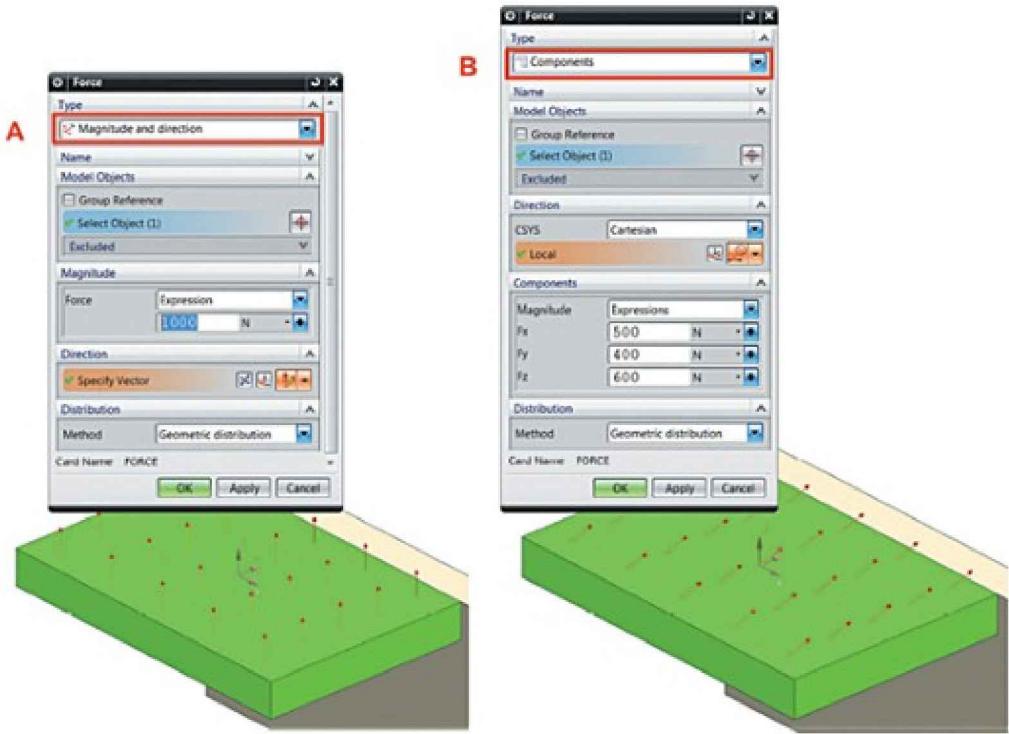


Figure 4.12. Specifying forces

You can use the **Bearing** command to specify the pressure applied to a cylindrical face or circular edge with a given vertex angle and a selected distribution law: sine or parabolic (Figure 4.13).

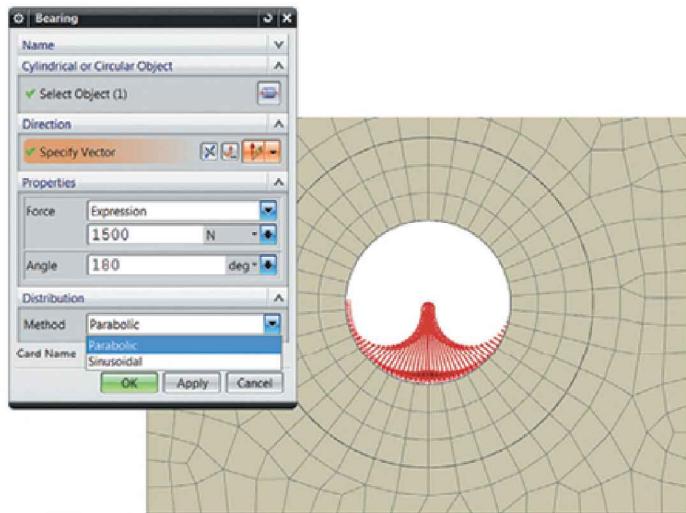


Figure 4.13. Specifying bearing

You can use the **Torque** command to simulate torques by applying

tangential forces to a selected cylindrical face. In this case the load is equivalent to the specified torque value (Figure 4.14).

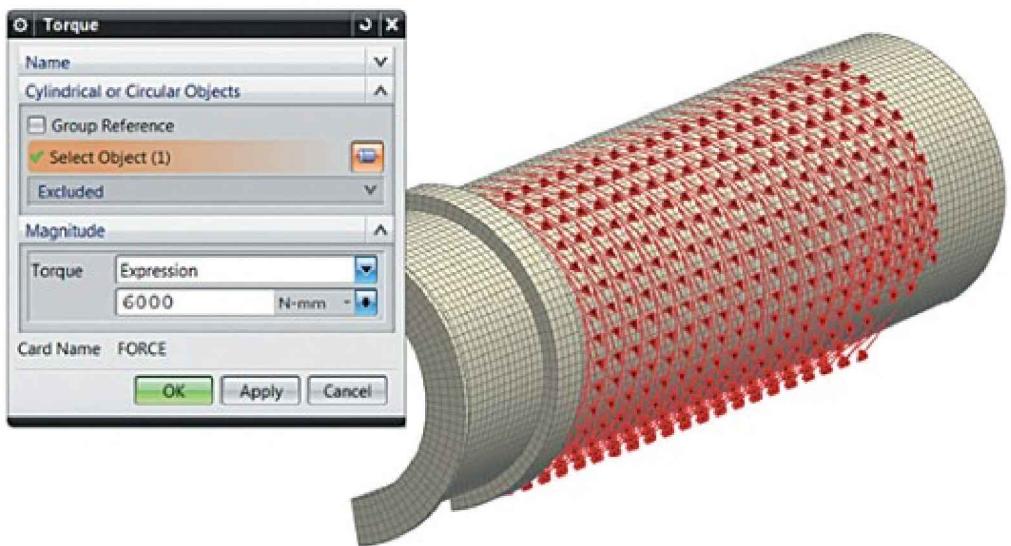


Figure 4.14. Specifying torque

You can use the **Pressure** or **Nodal Pressure** commands to apply pressure to faces with a constant value throughout an element or with different values at nodes of the element face. The options of the dialog boxes are identical (Figure 4.15):

- Select one of the following *Type* options for specifying values:
 - Select *Normal pressure on 2D elements or 3D elements faces* to apply the given pressure value to the selected faces.
 - Select *Normal pressure on 2D elements* to apply the given value to selected faces of 2D elements only.
 - Select *Components* to specify normal stress component values in the given coordinate system and apply these to selected objects.
 - Select *On beams* to specify normal stress component values relative to beam element coordinate systems and apply these to selected beam elements.
 - Select *Normal pressure on 2D elements or 3D elements faces – spatial*, *Normal pressure on 2D elements – spatial*, *Components – spatial*, or *On beams – spatial* in a way similar to the previous four types. However, the distribution type for

these types is always spatial.

- Specify objects, values, and if necessary, coordinate systems in accordance with the selected force application type.
- Select the *Distribution* method for the first four types: *Expression* or *Spatial*. For the rest of the types, the distribution method is always spatial, and you need to specify a data field.

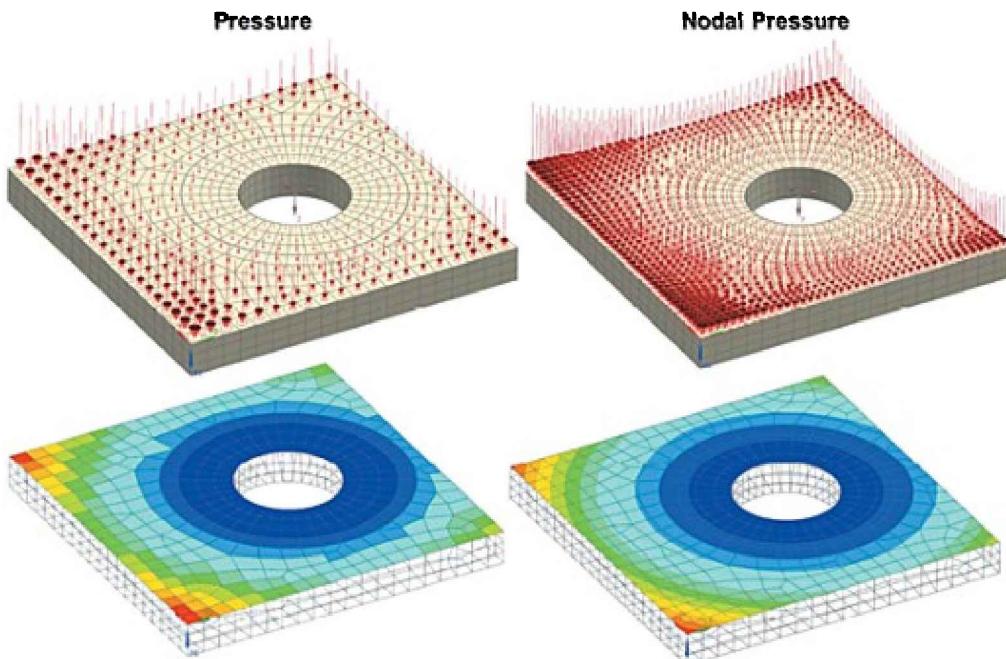


Figure 4.15. Specifying pressure

Select **Hydrostatic Pressure** to specify fluid pressure that varies with height in accordance with the specified gravitational constant, density, and free surface of the liquid.

Select **Centrifugal Pressure** to apply centrifugal pressure to selected objects. Centrifugal pressure occurs when a body filled with a fluid is rotated.

Select **Gravity** to set acceleration with a specified value, or using components, or for a selected direction. There is no need to select objects, the acceleration is supposed to affect the entire FE model.

Select **Centrifugal** to apply a centrifugal load that is caused by rotation of the body. Specify angular velocity, acceleration, and the direction. There is no need to select objects, the acceleration is supposed to affect the entire

FE model.

Select **Temperature Load** to specify a known temperature distribution in the object to process temperature-induced strain. Select one of the following Type options for specifying temperature values:

- Select *Temperature* to specify the temperature of selected objects.
- Select *Node ID Table* to use a data field containing node numbers and scale factors for the specified value.
- Select *Temperature – spatial* to specify the temperature of selected objects with spatial distribution only.
- Select objects and values in accordance with the selected temperature application type.
- Select *Distribution* method to use for objects.

Select **Bolt Pre-Load** to specify the load applied to bolt joints. Applicable only to CBAR and CBEAM rod-type finite elements.

Select **Axial 1D Deformation** to specify axial deformation values for 1D elements. Positive values correspond to elongation, negative values correspond to contraction.

4.3.2. Specifying constraints on degrees of freedom

In addition to loading conditions, you need to apply constraints on degrees of freedom to fully describe the simulation problem. It is known that each node of a finite element has a certain number of degrees of freedom (DOF) that unambiguously position the node in space when the FE model is deformed. A set or list of degrees of freedom of the model depends on the type of elements used in the simulation. For example, in nodes of the elements that work in bending and torsion (beams and shells), all six displacement components are defined (3 translational and 3 rotational), whereas in three-dimensional continuum element nodes only 3 translational displacements are defined. NX Nastran uses the following symbols for degrees of freedom:

- Translational *DOF1*, *DOF2*, and *DOF3* correspond to X, Y, and Z Cartesian directions respectively. *R*, *T*, and *Z* correspond to directions in a cylindrical coordinate system, and *R*, *T*, and *P*

correspond to directions in a spherical coordinate system.

- *DOF4*, *DOF5*, and *DOF6* correspond to rotation around the respective coordinate axes (Figure 4.5).

Similarly to loading, constraints can be applied to finite-element objects and geometry objects. In the latter case, when you save the solver file, constraints are transferred from geometry objects to the corresponding nodes considering nodal coordinate systems (para. 4.2.1). Note that when you select objects for a single command, they must be of the same type, for example, if you select some nodes, you cannot additionally select faces.

Consider the commands that you can use to set constraints on degrees of freedom to solve deformable solid body mechanics problems (Figure 4.16) in NX Nastran.

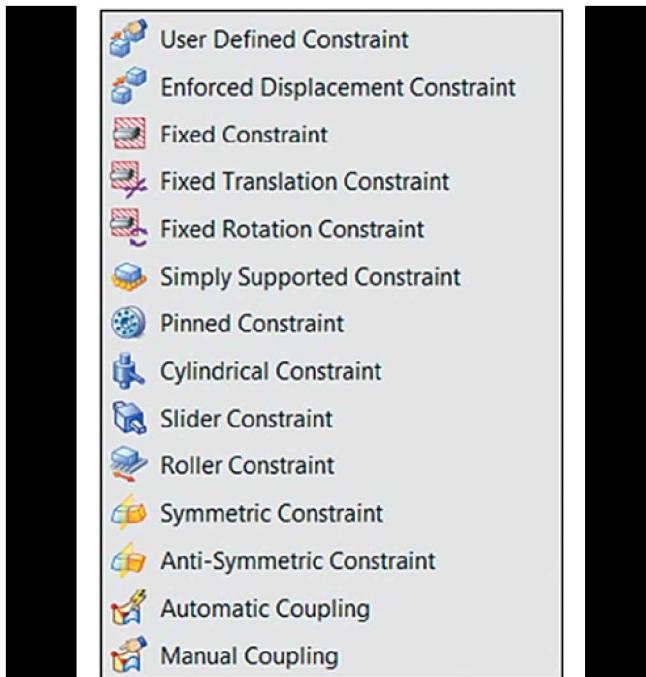


Figure 4.16. Types of constraints

Use **User Defined Constraint** to specify general constraints on degrees of freedom in the selected coordinate system. This command can cover all other constraint setting methods. Here is a description of the most important options of the dialog box (Figure 4.17):

- Select *Type* to choose the method used to record constraints in

the NX output file for NX Nastran (SPC or SPC1). The default option is typically SPC.

- Under *Model Objects*, select objects directly in the graphics area or specify an existing group or a group you create right in this dialog box.
- Under *Direction*, select the coordinate system that is used for the degrees of freedom.
- Under *Degrees of freedom*, set constraints on degrees of freedom as such:
 - Select *Free* if the degree of freedom is not constrained (default).
 - Select *Fixed* if the degree of freedom is constrained and has zero displacements and zero rotations – Select *Displacement* if the degree of freedom has a certain constant displacement value (enforced displacement constraint).
- Use *All Free* to reset all degrees of freedom to *Free*.
- Use *All Fixed* to set *Fixed* for all degrees of freedom.

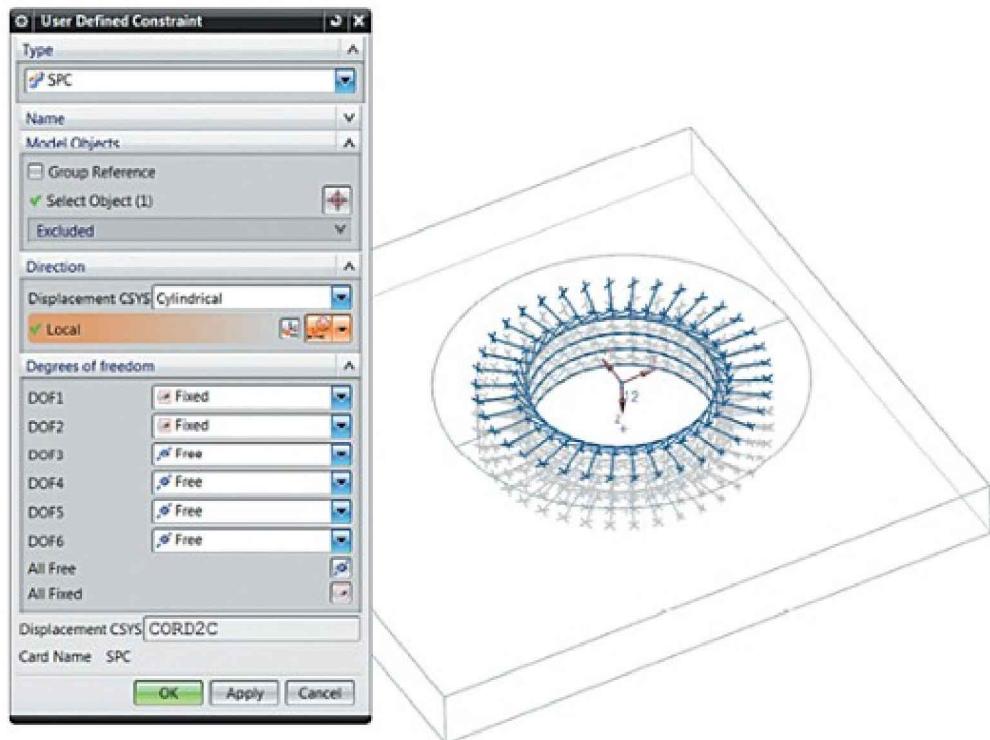


Figure 4.17. User defined constraint

Use ***Enforced Displacement Constraint*** to specify a constant (*Expression*) or variable (*Field*) displacement value for selected degrees of freedom of specified objects using a setup type:

- Select one of the following *Type* options for specifying values:
 - Select *Components* to specify constant or variable degree of freedom values for the specified coordinate system.
 - Select *Magnitude and direction* to specify a constant or variable magnitude and direction of the displacement vector.
 - Select *Normal* to apply displacement along the normal vector to the selected faces with a specified constant or variable magnitude.
 - Select *Magnitude and direction – spatial* to specify a spatial data field as the displacement magnitude with a scale factor and displacement direction vector.
 - Select *Normal – spatial* to apply displacement along the normal vector to the selected faces with a given spatial distribution data field.
 - Select *Translational components – field* to specify spatially variable translational degrees of freedom and corresponding scale factors for the selected coordinate system (using the given data field).
- Select the *Distribution* method for the first two types: *Expression* or *Spatial*. For the rest of the types the distribution is always special, and you need to specify a data field.

Use ***Fixed Constraint*** to fix all six degrees of freedom of the selected objects.

Use ***Fixed Translation Constraint*** to fix only three translational degrees of freedom of the selected objects.

Use ***Fixed Rotation Constraint*** to fix only three rotational degrees of freedom of the selected objects.

Use ***Simply Supported Constraint*** to specify a simple support condition by defining a vector to fix the corresponding degree of freedom and leaving the others free.

Use **Pinned Constraint** to create a nodal cylindrical coordinate system for the selected cylinder body. All degrees of freedom are fixed relative to this coordinate system except circumferential displacements and possible rotational degrees of freedom that are perpendicular to the rotation axis.

Use **Cylindrical Constraint** to create a local cylindrical coordinate system for the selected cylinder body. Constraints are set relative to this coordinate system for radial, circumferential, and axial components of the displacement vector (Figure 4.18).

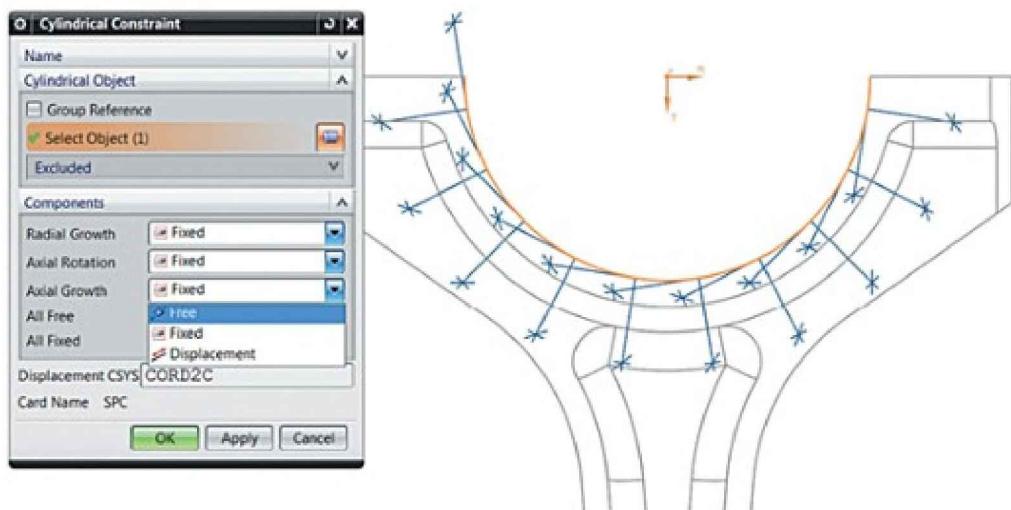


Figure 4.18. Cylindrical constraint

Use **Slider Constraint** to define a vector along which the degree of freedom is free, while all others are fixed.

Use **Roller Constraint** to select a vector along which one translational and rotational degrees of freedom are free, and all others are fixed.

Use **Symmetric Constraint or Anti-Symmetric Constraint** to set symmetric and anti-symmetric constraints respectively on faces of the plane of symmetry.

Use **Automatic Coupling** to automatically set coupling conditions on degrees of freedom between nodes of symmetrical or cyclically symmetrical meshes. For example, for cyclically symmetrical problems, you need to establish the equality of nodal degrees of freedom on sector boundaries correspondingly. When you set these conditions, you use concepts such as

independent nodes and dependent nodes, whose degrees of freedom depend on the degrees of freedom of independent nodes. If you set additional conditions on constraints, these must be applied to the independent nodes only. The software automatically finds the corresponding mesh nodes and couples the appropriate degrees of freedom. (Figure 4.19):

- Under *Independent Node*, select a single node or point on an independent face.
- Under *Dependent Node*, select the corresponding dependent node or point.
- Under *Reference Coordinate System*, specify the coordinate system for finding the corresponding nodes.
- Select *Set Displacement CSYS* to choose whether to assign the specified coordinate system as the nodal coordinate system for nodes that have coupling conditions applied.
- Under *Node Match Tolerance*, enter the tolerance for finding dependent nodes.
- Under *Degrees of Freedom*, select degrees of freedom to set coupling conditions for.

Use **Manual Coupling** to specify the degree of freedom coupling equation between nodes manually for the given type of coupling conditions:

- *Coupled DOF* corresponds to the equality of degrees of freedom of a single independent node and several dependent nodes.
- *MPC* allows to create coupling equations for any degrees of freedom and for any nodes, for example $2*DOF2(\text{node } 54)+5*DOF3(\text{node } 67)-3*DOF2(\text{node } 24)=0$.

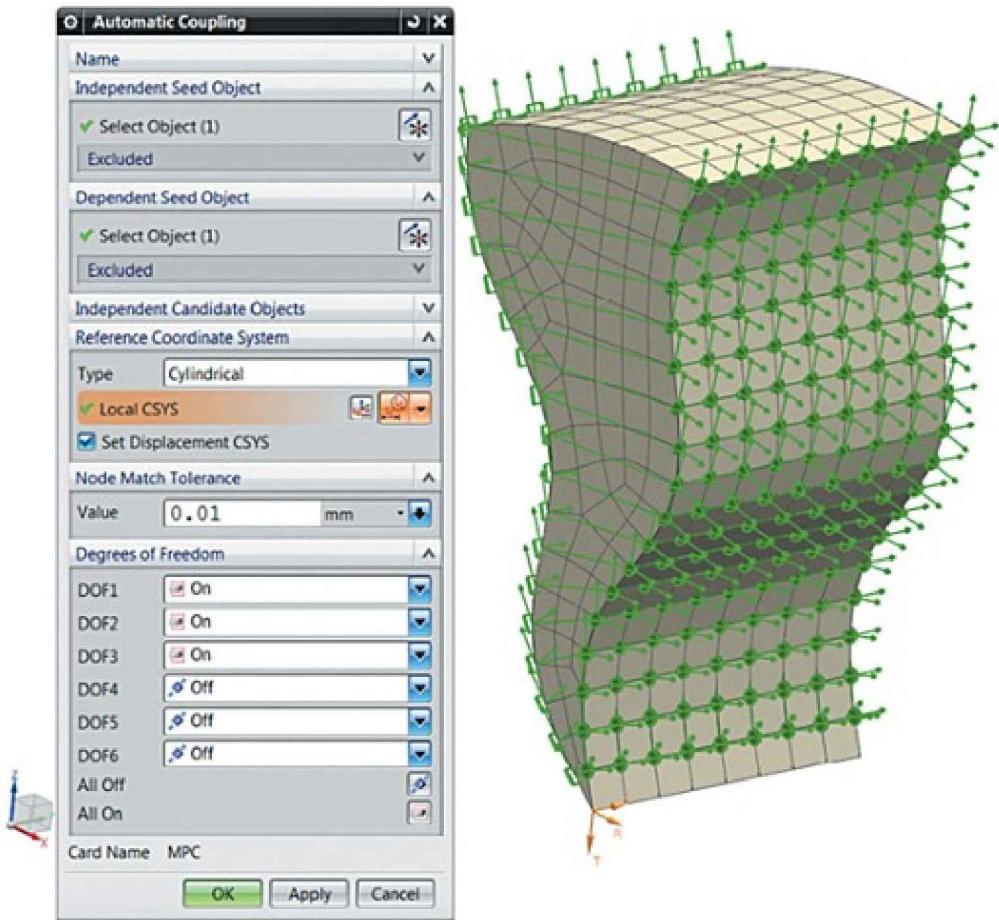


Figure 4.19. Coupling of degrees of freedom

You can use more constraints for some solution types. For example, in dynamic simulation you can set initial conditions, velocities and accelerations using the corresponding commands. The dialog boxes are not significantly different to the enforced displacement command.

4.3.3. Simulation objects, contact interaction

In addition to loads and constraints, there is another type of boundary condition: the so-called simulation objects responsible for surface-to-surface contact interaction, surface-to-surface and edge-to-surface gluing conditions, setting “element birth/death” conditions, and specifying initial temperature distribution for unsteady state thermal conductivity problems. In this section you can find a description of simulation objects for the FE model that can be used for solving deformable solid mechanics problems with NX

Nastran. You can call the commands using:

- The **Advanced Simulation** toolbar in the **Simulation Object Type** command list.
- The **(Simulation Navigator)** model tree view. Right-click the **Simulation Object Container** node and choose **New Simulation Object** in the shortcut menu.

Use **Surface-to-Surface Contact** to specify contact interaction conditions of body surfaces with subsequent nonlinear static or dynamic analysis (Figure 4.20). To create this conditions you can specify contact pairs. A contact pair consists of a contact surface (contact region) and a target surface (target region).

You can specify surface contact pairs by specifying the so-called regions that contain surface sets and additional options. You can assign regions for contact interaction in two ways:

- Use *Manual* to select regions from a list or create new regions.
- Use *Automatic Pairing* to automatically find surface pairs among user-defined surfaces and automatically create regions. Surfaces in close proximity are identified in the specified search distance. To set contact interaction conditions between surfaces that are fairly distant from each other, use the manual method.

To create or edit regions, use the **Simulation Region** command and set the following options:

- Select the *Type* of the regions (areas):
 - Select *Surface Region* to specify faces.
 - Select *Edge Region* to specify edges. You can use this option to specify gluing of an edge to a surface or in case of an axisymmetric problem.
- Select *Model Objects* to choose polygon faces/edges to include in the region.
- Under *Name*, enter the name and number of the region. The next available number is used by default.
- Under *Common Contact Parameters*, specify geometry-related parameters:

- Specify the *Surface* (top or bottom) of shell-type elements that will participate in the contact interaction. The top surface faces in the positive direction of the element normal vector, while the bottom surface faces in the negative direction.
- The *Offset* is specified only for the contact region. It is used in discovering contact in case of nonzero gap between the objects and a specified value.
- Set the *Nonlinear Contact Parameters* to specify whether the region is to be considered absolutely stiff. It is meaningful only for nonlinear SOL 601 and SOL 701 solutions.

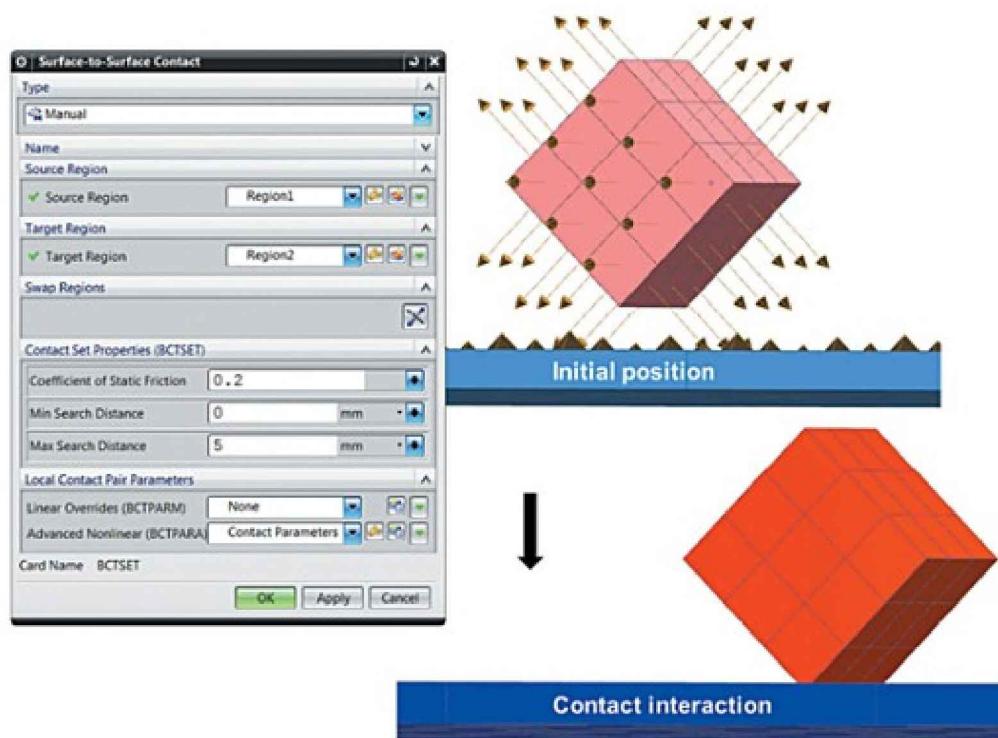


Figure 4.20. Contact interaction

After using the **Surface-to-Surface Contact** command to create contact interaction condition, set up the following options (Figure 4.20):

- Select *Type* to choose the region assignment method (automatic or manual).

If you select the automatic method, you need to specify objects, create face pairs, and assign *Contact Region Parameters*:

- Enter the *Source Offset* to specify the *Offset* of the contact regions.
 - Enter the *Target Offset* to specify the *Offset* of the target regions.
 - Under *Source Contact Side*, specify the shell element surface for contact regions (only applicable to shell elements).
 - Under *Target Contact Side*, specify the shell element surface for target regions (only applicable to shell elements).
 - Select *Target Region Type*: absolutely stiff or not.
- In the *Contact Set Properties* group, specify static friction coefficient and contact search range.
- In the *Local Contact Pair Parameters* group, set up options for modeling “linear” and nonlinear contact interaction.
- *Boundary Condition Generation* works only for automatic method of region assignment. When you select the check box and modify the geometry, the software recreates the pairs and updates contact interaction conditions.

Use **Surface-to-Surface Gluing** to specify surface-to-surface gluing conditions to prevent relative movement in all directions (Figure 4.21). The method of creating these conditions is completely analogous to the method of creating contact interaction conditions with fewer parameters. It is supported by all NX Nastran solutions except SOL 701 and axisymmetric analyses. It can be used to connect 3D element faces, 2D element faces, and faces of 3D elements with 2D elements.

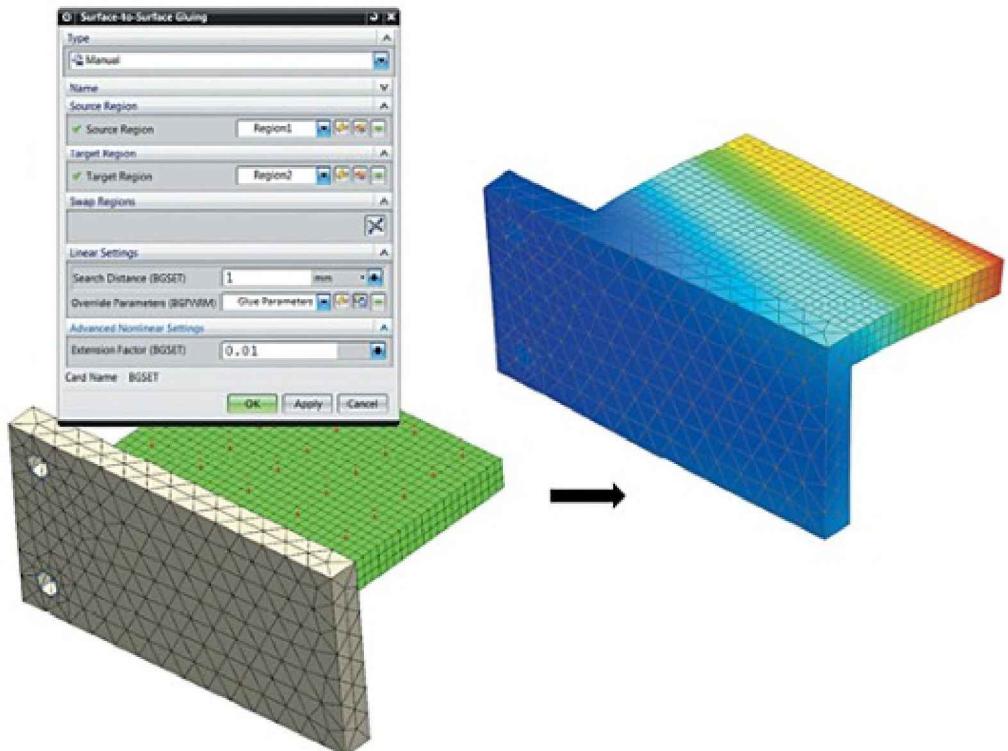


Figure 4.21. Surface-to-surface coupling condition

Use **Edge-to-Surface Gluing** to specify edge-to-surface gluing conditions and prevent their relative movement in all directions (Figure 4.22). It is supported by all NX Nastran solutions except SOL 601, SOL 701, and axisymmetric analyses. These conditions are typically used to couple edges of shell elements to faces of 3D elements. To set up an edge-to-surface coupling condition, specify the following parameters:

- Under *Source Region*, specify a region that consists of 2D or 3D element edges.
- Under *Target Region*, specify regions of surfaces that can correspond to 2D and 3D elements.
- Under *Linear Settings*, specify parameters for maximum tolerance between objects, and *Override Parameters* that you can use to manage compliance at the coupling boundary.

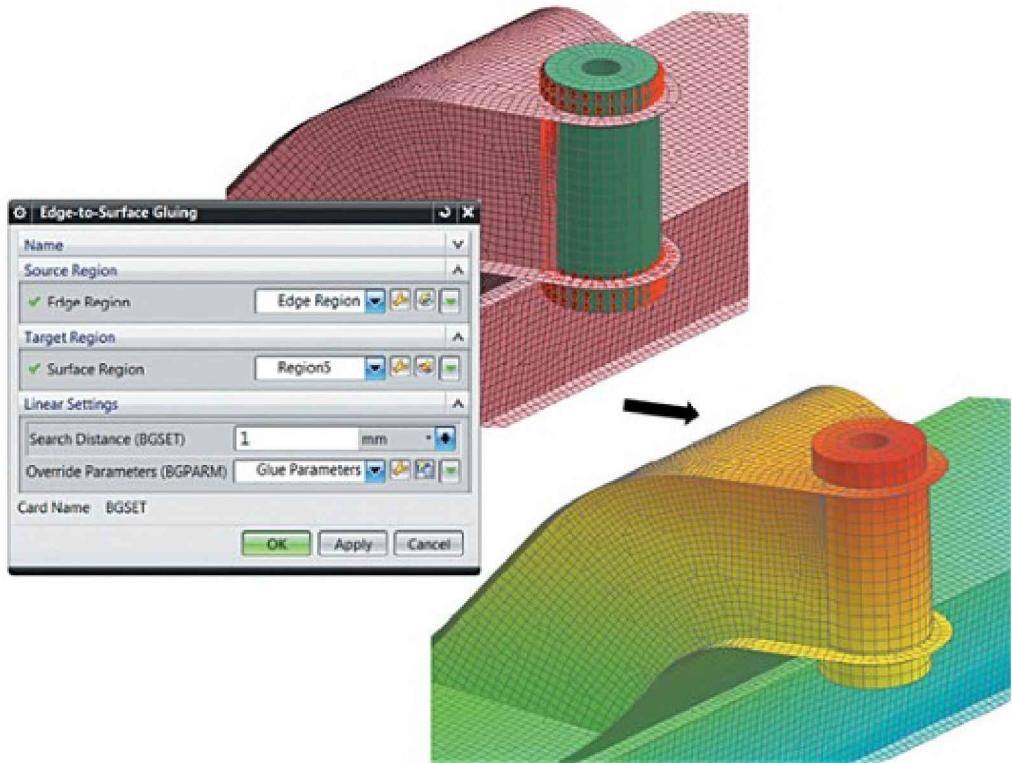


Figure 4.22. Edge-to-surface coupling condition

Use **Initial Temperatures** to specify the temperature distribution in the model and use it in an unsteady thermal conductivity problem as the initial conditions. The methods of applying temperatures to the model are analogous to specifying loading conditions (. 4.3.1).

Use **Element Birth/Death** to specify conditions used with the element “birth/death” technology. When you perform nonlinear static or dynamic analysis, you can introduce or withdraw elements at any time. You simply need to select the elements and specify “birth” and “death” times.

4.4. Preparing the solution

After preparing the FE simulation model in the **NX Advanced Simulation** pre-processor, you need to simulate the problem in NX Nastran or some other numerical analysis system. When you run the simulation from the pre-processor, the software creates an input file to calculate the selected solution and processes it in the solver (by numerically solving the differential equation system).

To create the required solution and specify analysis type, right-click the simulation model name in the model tree of the **Simulation Navigator** tab and choose **New Solution**. In the dialog box, type the name of the new solution, select *Solver*, *Analysis Type*, and *Solution Type*. If necessary, override default solution parameters (Figure 4.23).

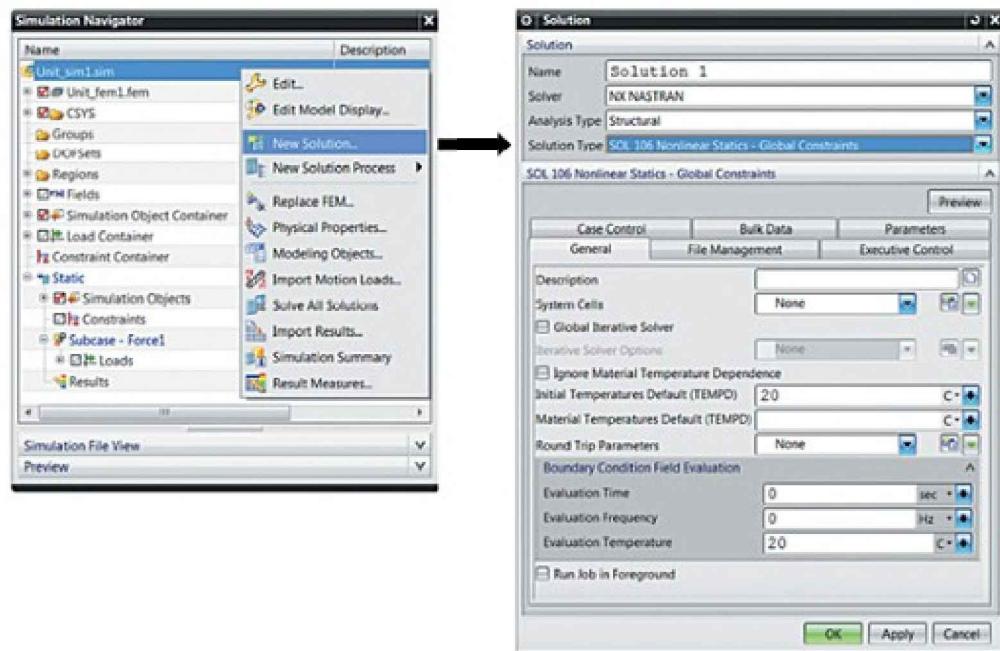


Figure 4.23. Creating a solution

The newly created solution is filled with simulation model objects in the simulation model tree. To modify solution parameters, right-click the solution name and select **Edit** in the shortcut menu. Each solver has its own set of solution parameters, therefore they are not discussed in this chapter. Some of them are treated in subsequent chapters that concern different analyses and solutions. You can also set solver parameters using the **Edit Solver Parameters** command and specify the executable file of the solver, RAM allocation, temporary folder assignment, and so on. You can also delete, rename, or duplicate solutions using other commands.

If you use several loading types in a single solution, you can manage simulation model objects between different simulation events using the **Subcases Manager**.

Note that a single simulation model can include different types of analyses

and solutions. For example, you can create: a NX Thermal thermal conductivity solution, a NX Nastran linear static NX Nastran problem solution, a nonlinear dynamic analysis, a durability analysis, and so on.

To simulate the simulation model, right-click the solution name in **Simulation Navigator** and choose **Solve....** Alternatively, on the **Advanced Simulation** toolbar, you can select the **Solve** command. It is recommended that in the **Solve** dialog box, you select the *Model Setup Check* check box to eliminate errors in the simulation model. This option allows to avoid errors like failing to specify the material, completely failing to specify boundary conditions, incompletely specifying physical properties, and so on . You can also set up solution parameters, solver parameters, and simulation model file creation options for the solver before you run the simulation.

Note that when you run the solution using NX solvers (NX Nastran, NX Thermal/Flow and so on), the so-called **Analysis Job Monitor** opens, which you can use it to monitor completed solutions, as well as the **Solution Monitor**, which displays the solution process in real time. You can use the information displayed as plots in separate tabs of the solution monitor to evaluate the simulation stage or convergence of algorithms in nonlinear problem, edit preset dynamic analysis values, and perform many other activities (Figure 4.24). Analyzing plots allows evaluating the speed of solution and convergence of problems. This data can lead to realization that the parameters of the simulation model need to be changed to improve solution performance.

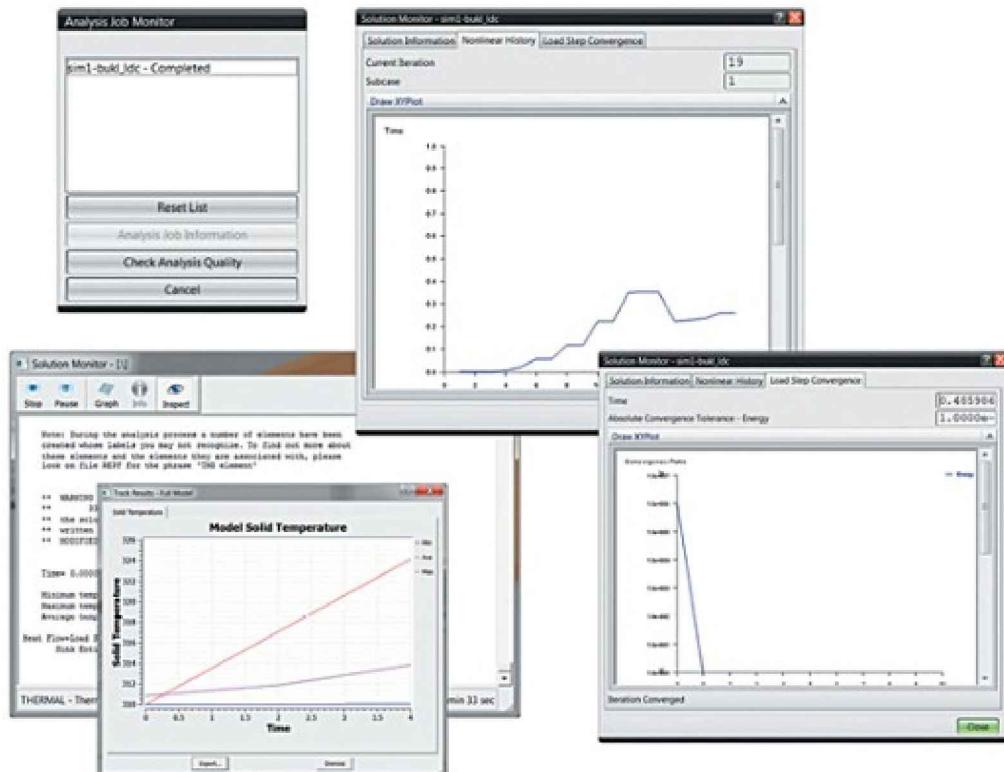


Figure 4.24. Solution Monitor

When the simulation is completed, a *Completed* message appears in the analysis monitor, and a *Job Finished* message appears in the solution monitor. In the completed solution node of the **Simulation Navigator** tab model tree, the *Results* node appears. Double-click it to select the **Post Processing Navigator** tab with results loaded.

4.5. Example. Creating and solving a simulation model

In this example you will create a simulation model for statistical analysis of a U-shaped lifting gear frame working in lifting mode (Figure 4.25). You will use the finite-element model created in the example of Chapter 3. To demonstrate **NX Advanced Simulation** capabilities, you need to develop a simulation model.

Engineering analysis involves several steps:

1. Opening the FE model and creating a new simulation model file.
2. Setting conditions on contact interaction and gluing for parts of the

structure.

3. Specifying loads.
4. Specifying constraints on degrees of freedom.
5. Performing the static analysis.
6. Viewing static analysis results.

In this example almost all commands are invoked from the **Advanced Simulation** toolbar, where the commands are grouped logically. To open this toolbar (if it is not already displayed), right-click in any place except the graphics area and choose the required toolbar in the available toolbar list to display it.

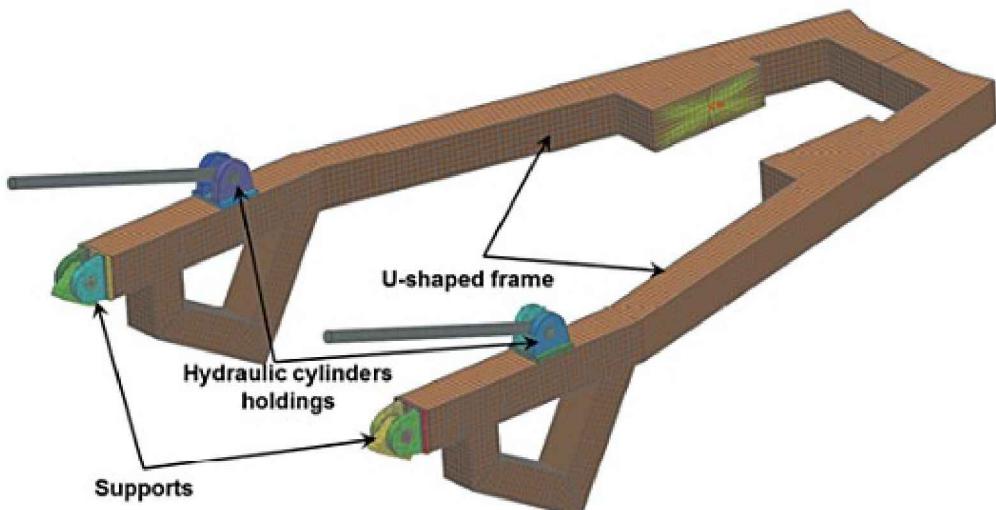


Figure 4.25. FE model of the U-shaped frame

The initial FE model is a collection of shell elements that describe the thin-shell part of the structure, and a collection of spatial solid-body elements that are created by splitting hydraulic cylinder fixtures and hinged bearing fixtures. Gluing conditions are set up between shell elements and spatial elements to correctly transmit force and kinematic constraints. The operation of cylindrical parts in fixtures is achieved using contact interaction conditions. The frame is loaded by its weight when being lifted.

4.5.1. Opening the FE model and creating a new simulation model file.

Launch NX and open the *Frame_fem1.fem* file, then go to NX Advanced Simulation (*Start→Advanced Simulation*).

Leave default settings in all dialog boxes. In the main menu choose: **Preferences**→**User Interface**→**General**→**Reset Dialog Box Settings**, click **OK**.

Right-click the FE model in **Simulation Navigator** or in the **Simulation File View** dialog box (Figure 4.26). Select **New Simulation** to create a simulation model. In the simulation file creation dialog box, enter the name and folder of the new file, make sure *NX Nastran* is selected as the template, and click **OK**. In the simulation file creation dialog box, click **OK**. In the new dialog box you can select a solver and a new solution. Make sure that in the **Solution** dialog box under **Solution Type**, you select *SESTATIC 101 – One constraint*, and click **OK**.

In this way you create a simulation model based on the source FE model. In the **Simulation File View** the simulation model file is active. Note that the simulation model file is populated with empty containers that will be filled when the actual simulation model is created.

4.5.2. Setting conditions on contact interaction and gluing for parts of the structure.

The current FE model contains 2D shell and 3D solid-body finite elements that are not connected to each other. To connect the mesh and correctly describe the interaction of movable parts in the structure, you need to specify simulation objects that are responsible for surface-to-surface contact interactions, as well as surface-to-surface and edge-to-surface gluing conditions.

Hide 2D and 3D meshes helps to make selecting objects for commands easier.

To create gluing conditions for surfaces of shell elements of the U-shaped frame with 3D solid-body elements around hydraulic cylinders fixtures, use the **Surface-to-Surface Gluing** command, which you can find in the **Simulation Object Type** shortcut menu of the Advanced Simulation toolbar. In the dialog box, do the following (Figure 4.26):

- For *Automatic Pairing* type, click the **Automatic Face Pair Creation** command.
- In the new window, select two faces of the frame and one face

from each of the two bodies that contact the frame as shown in Figure 4.26.

- There is a distance of 5 mm between the relevant faces, therefore set 10 mm as the *Distance Tolerance* value.
- Click **OK**.
- Specify 10 mm as the *Search Distance*.

Click **OK**.

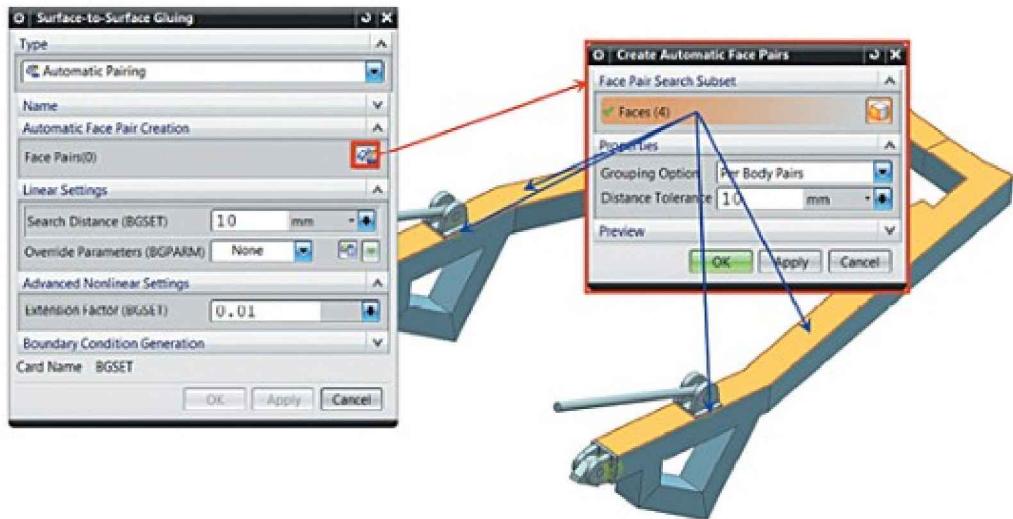


Figure 4.26. Setting surface-to-surface gluing conditions

To create gluing conditions for edges of shell elements of the U-shaped frame with surfaces of 3D solid-body elements around hinged support fixtures, use the **Edge-to-Surface Gluing** command, which you can find in the **Simulation Object Type** shortcut menu of the **Advanced Simulation** toolbar. In the dialog box specify the following (Figure 4.27):

- **Source Region:** use the **Create Region** command and select all midsurface edges that contact edges of the left hinged support. Click **OK**.
- **Target Region:** use the **Create Region** command and select the face that contacts previously specified edges. Click **OK**.

Click **OK**.

Create edge-to-face gluing conditions for the right hinged support on your own.

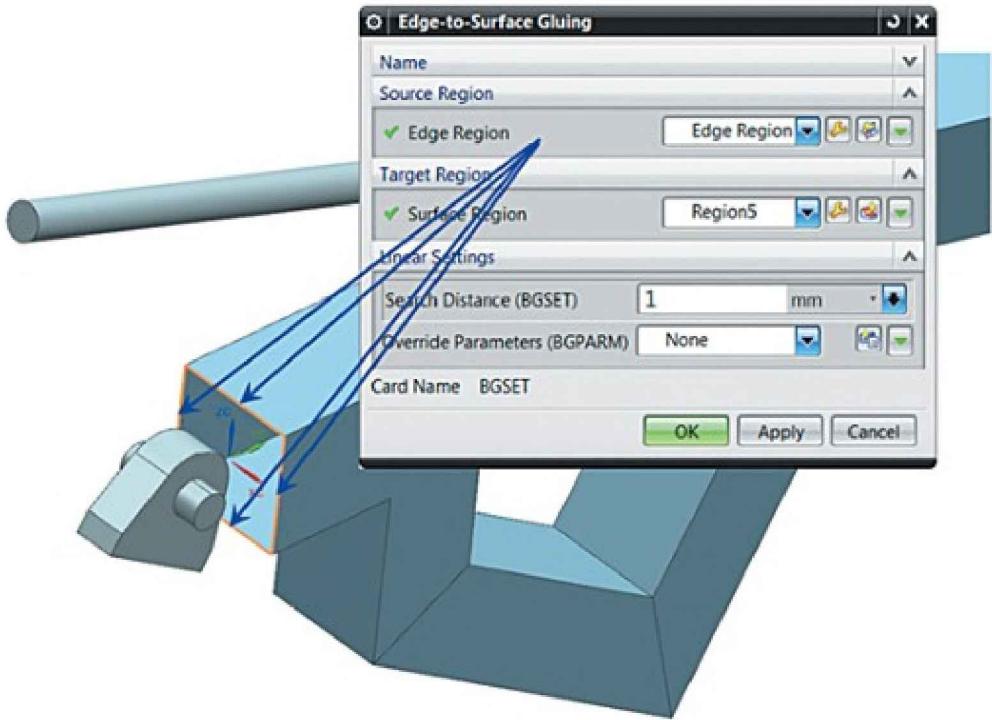


Figure 4.27. Setting edge-to-surface gluing conditions

Use the **Surface-to-Surface Contact** command. Choose it in the **Simulation Object Type** shortcut menu of the **Advanced Simulation** toolbar. In the dialog box do the following:

- For **Automatic Pairing** type, click the **Automatic Face Pair Creation** command.
- In the new dialog box, select all body faces. Only touching faces that do not participate in previously created connections will be found.

Click **OK** in all dialog boxes.

4.5.3. Specifying loads

Before you specify loads, you need to create a local coordinate system that will be used for applying forces. Use the **Orient WCS** command on the **Utility** toolbar and set the following options (Figure 4.28):

- Under **Type**, select **Offset CSYS**.
- Specify the **Translation**: $X = 1920 \text{ mm}$, $Y = 6394.545 \text{ mm}$, $Z = 0$.

- Specify the *Rotation*: X angle = -82 deg, Y angle = 10 deg, Z angle = 0 deg.

Click **OK**.

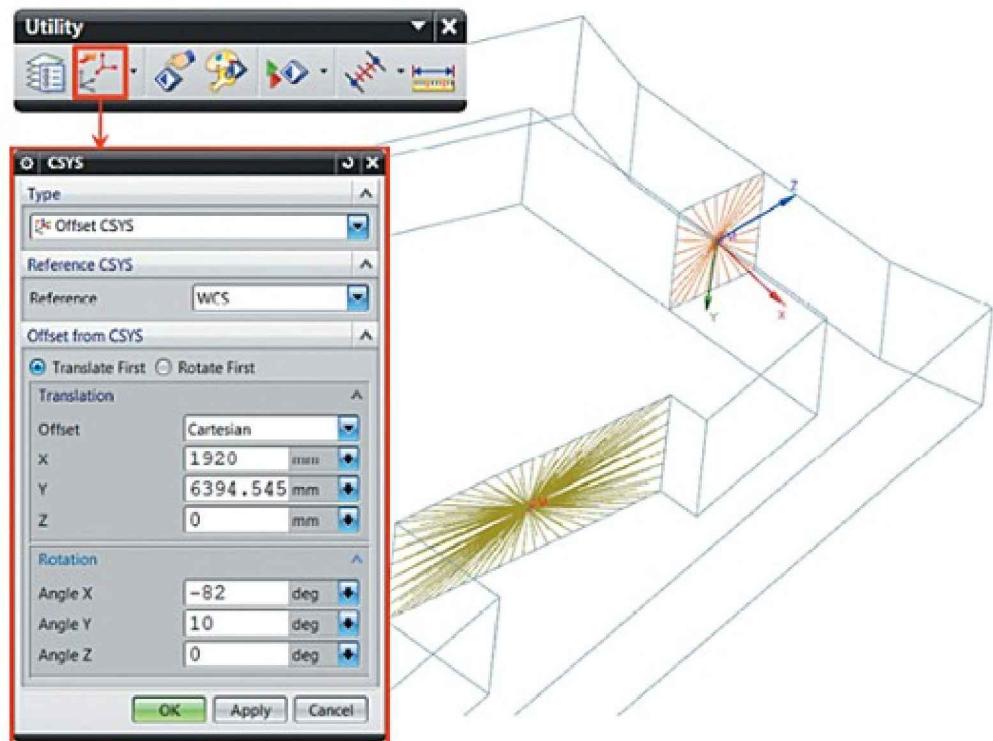


Figure 4.28. Creating a local coordinate system

Apply the force that arises when the frame lifts a load. To do so, in the **Load Type** shortcut menu (**Advanced Simulation toolbar**), choose **Force**. In the dialog box, specify the following (Figure 4.27):

- Under *Type*, select *Components*.
- Use *Select Object* to select the node corresponding to a point mass of 500 kg, as shown in Figure 4.29.
- Under *CSYS*, select *Cartesian*. In the graphics window, specify the local coordinate system you just created.
- Enter *Components* values: $F_x = 0 \text{ N}$, $F_y = 12000 \text{ N}$, $F_z = -11000 \text{ N}$.

Click **OK**.

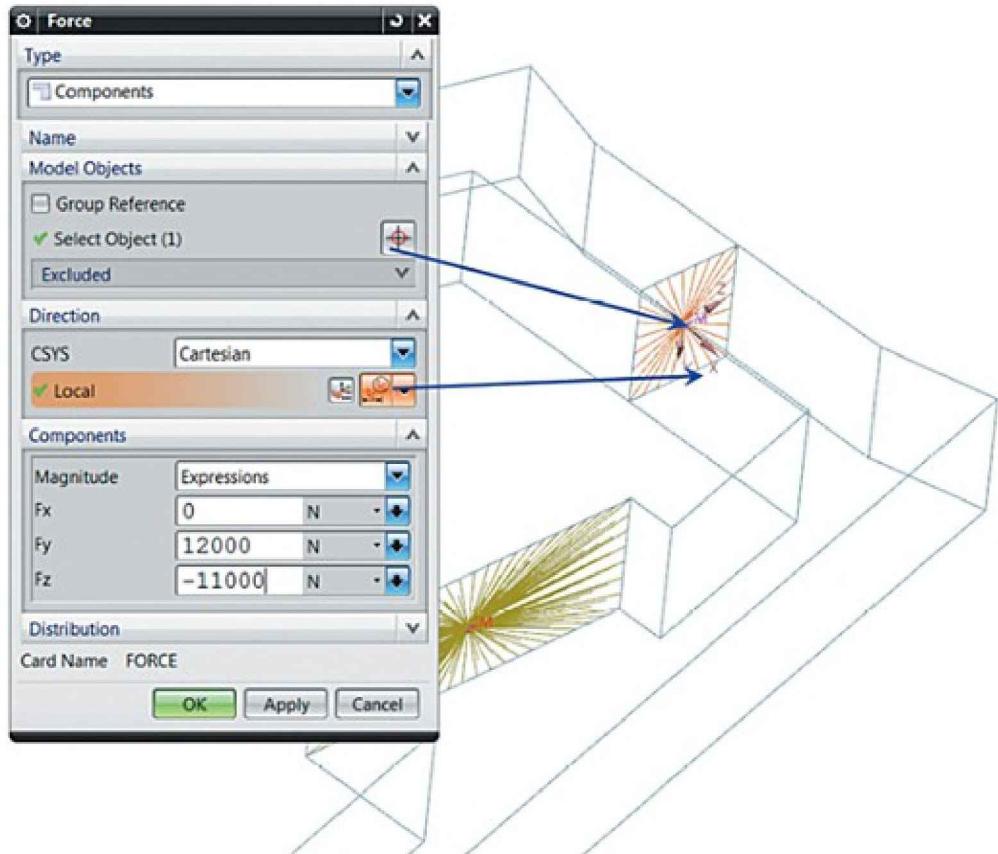


Figure 4.30. Specifying a point force

To specify a gravity load, choose **Gravity** in the **Load Type** shortcut menu (Advanced Simulation toolbar).

In the dialog box, specify the gravity force direction by using *Specify Vector* and selecting the edge of the support as shown in Figure 4.30. Make sure you specify the direction correctly. Otherwise, click *Reverse Direction*. Click **OK**.

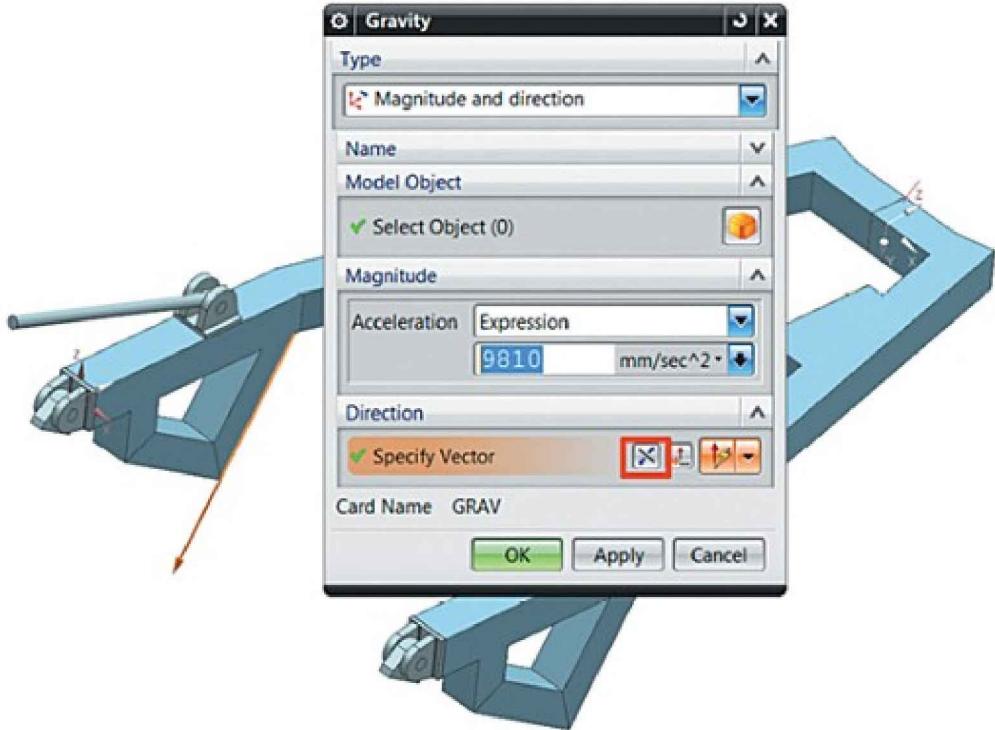


Figure 4.30. Specifying gravity forces

4.5.4. Specifying constraints on degrees of freedom

Now that all loading conditions and simulation objects are set up, you need to set the movement constraints.

Choose the **Fixed Constraint** command from the **Constraint Type** shortcut menu of the **Advanced Simulation** and select one face on each of the hydraulic cylinders and hinged supports, as shown in Figure 4.31. Click **OK**.

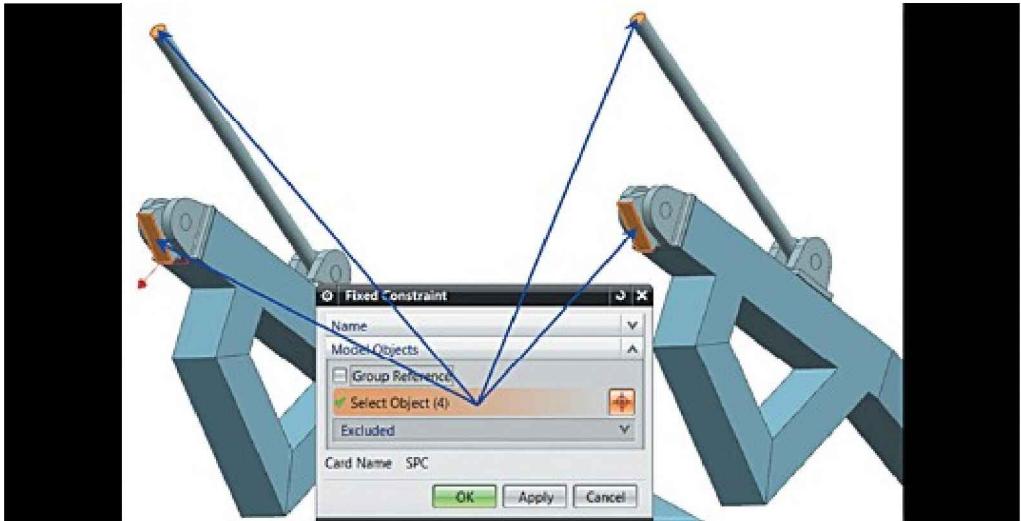


Figure 4.31. Constraining all degrees of freedom

Since in this example cylindrical bushings are in contact interaction with eye rings, for the sake of simplicity possible displacement of bushings along their axes can be neglected to improve convergence of the problem's solution. In the **Constraint Type** shortcut menu of the **Advanced Simulation** toolbar, choose the **User Defined Constraint** command, select one end face per cylindrical bushing, and under *DOF1* select *Fixed*. Click **OK**.

4.5.5. Performing the static analysis

The **Simulation Navigator** model tree shows containers of loads, constraints, and simulation objects. Expand them and make sure they are set up correctly. The simulation model is ready for simulation. Save the model. To do this, in **Simulation Navigator** right-click *Frame_fem1_sim1* in the **Simulation File View** window and choose **Save**.

To solve the simulation model you created, right-click *Solution 1* and choose **Solve....** Click **OK**.

Note that multiple windows appear. The **Information** window contains information about the simulation model and verification data. The **Solution Monitor** window contains information concerning the solution progress. The **Sparse Matrix Solver** tab of this window shows the number of equations solved up to now. The **Contact Analysis Convergence** tab shows the convergence process of the linear contact interaction conditions algorithm. When the *Completed* message appears in the bottom part of the analysis

monitor window, and the *Job Finished* message appears in the solution monitor, close all new solution-related windows.

4.5.6. Viewing static analysis results

In the **Simulation Navigator** model tree, double-click the *Results* of the active solution *Solution 1* to use the **Post Processing Navigator** with results loaded. To view the results, select the *Solution 1* tab, and click *Nodal Displacement*. The graphics area shows the distribution of total displacement (displacement vector modulus) over nodes.

To hide finite element boundaries, use the **Edit Post View** command (**Post Processing toolbar**). In the **Post View** dialog box, select the *Edges & Faces* tab, and under *Primary Display – Edges*, select *None* (Figure 4.32). Click **OK**.

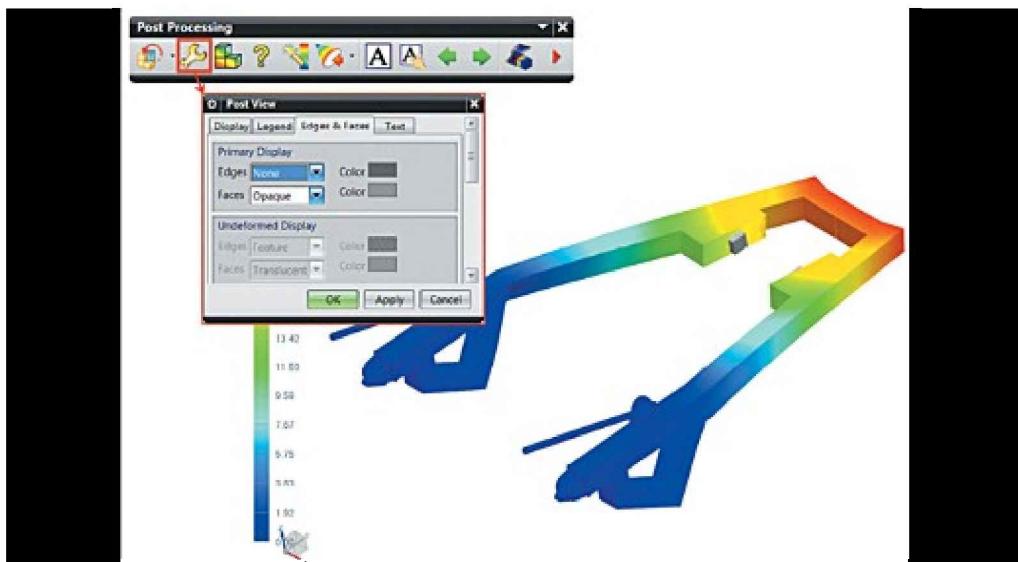


Figure 4.32. Setting up model display preferences in the postprocessor

Display the von Mises equivalent stress distribution. To do this, double-click the *Stress – Element Nodal* node of the **Post Processing Navigator** tab.

To display the minimum and maximum value of the selected parameter, in the **Post Processing** toolbar, click **Marker On/Off** (Figure 4.33).

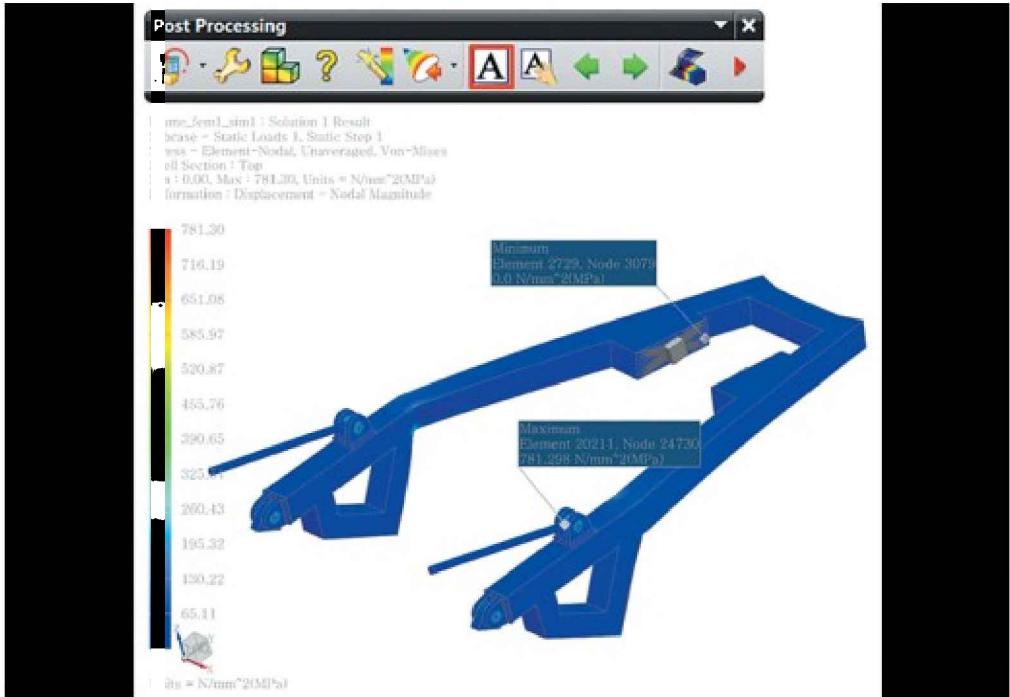


Figure 4.33. Displaying the minimum and maximum values of a selected parameter

You can use the displayed results to find out an unacceptable stress level (780 MPa) at hydraulic cylinder fixtures, at hinged supports, and in the bends of the U-frame. Therefore, you need to modify structure, for example, add stiffener ribs to the frame, or increase the thickness of the frame's sheet steel. When you modify the thickness of the frame, you do not need to create new geometry and simulation model. You can simply change the thickness of shell elements in the physical property table and repeat the simulation. Try to introduce this modification on your own and compare the results.

When you are done working with the model, close it by choosing **File→Close→All Parts** in the main menu.

Chapter 5. Results visualization and post processing tools

Correct interpretation of engineering analysis results is very important. NX **Advanced Simulation** postprocessor provides a broad range of tools for results visualization and processing.

You can view, manage, and format numerical analysis results using the following capabilities of the postprocessor:

- Displaying results as contour plots with a broad selection of visualization tools.
- Reading and processing numerical values.
- Representing results in graphical form.

You can also use some additional features:

- Animating results as a video sequence.
- Flexible selection of results for saving in a file.
- Support for BMP, JPG, WMF, AVI, and VRML formats.
- Creating JT files.
- Representing results in tabular form.
- Saving results in an MS Excel spreadsheet.

You can manage the result files in the **Post Processing Navigator** tab, and all basic result processing tools are available on the **Post Processing toolbar**.

5.1. Post Processing Navigator

5.1.1. Post Processing Navigator results tree

The **Post Processing Navigator** tab shows an interactive hierarchical tree of result files (the results tree). This tab has a table layout with two columns, *Name* and *Description*. Click the right mouse button in the top of the column to access the shortcut menu so that you can use to *Collapse All* expanded nodes, *Expand All* nodes, *Export to browser*, or *Export to Spreadsheet* (Figure 5.1).

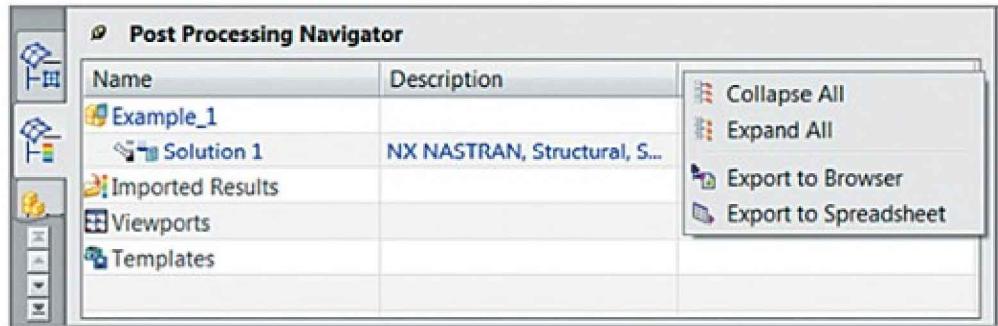


Figure 5.1. Post Processing Navigator tab

The names of all simulation models loaded in the **Simulation Navigator** tab (the names of the open S/M files) are displayed in the **Post Processing Navigator** tab. The results tree bullet that corresponds to the active simulation model is highlighted in blue, while bullets of inactive simulation models are displayed in gray. If there are no solutions in the simulation model, the results tree contains only one bullet, which has the same name as the S/M file. As soon as a new solution is created in the simulation model, a new object with the name of the new solution is added to the corresponding bullet of the results tree (Figure 5.2).

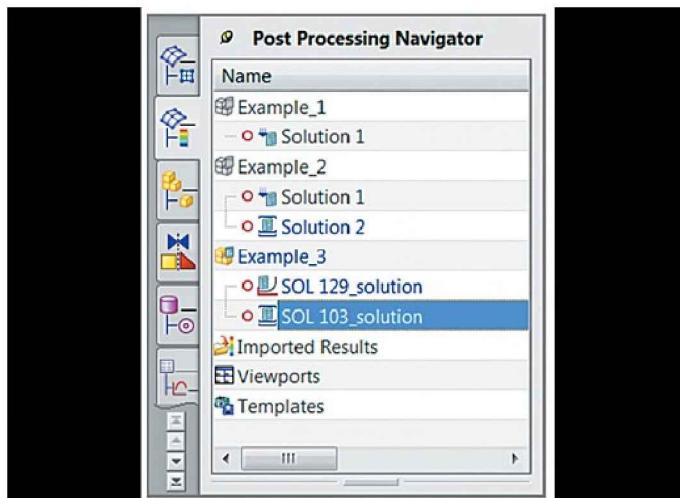


Figure 5.2. Post Processing Navigator tab

In addition, all solutions in the **Post Processing Navigator** tab have indicators that mark their current status:

- Unfilled red circle: the results file is not loaded, or the corresponding solution was not calculated (Figure 5.3, A).

- Grey “palette ruler”: the simulation is complete (the results file is loaded) but inactive (Figure 5.3, B).
- Coloured “palette ruler”: the simulation is complete, and the solution is active (Figure 5.3, C).

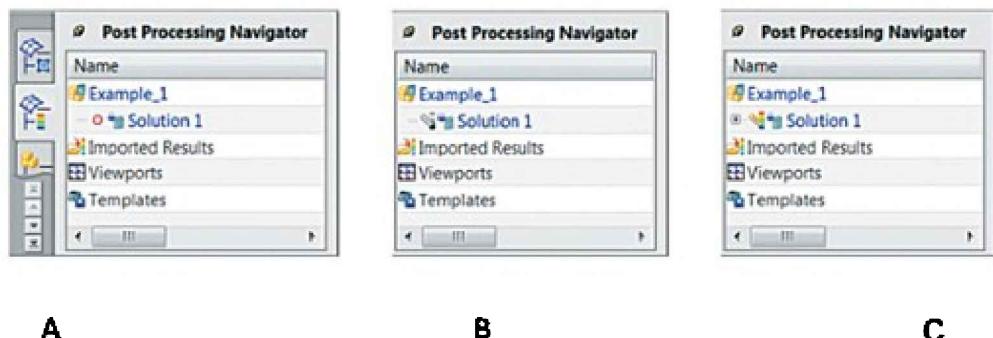


Figure 5.3. Post Processing Navigator tab:

- A. *Simulation not completed.* B. *Simulation completed but inactive.* C. *Simulation completed and active*

As soon as a satisfactory solution to the problem is found, a results file with the op2 extension is created. You can load the results for display and analysis using one of the following methods:

- In the model tree view of the **Simulation Navigator** tab, double-click the *Results* node. You are then automatically taken to the **Post Processing Navigator** with the results loaded (Figure 5.4, A).
- In the **Post Processing Navigator** tab, double-click the item with the name of the completed solution (Figure 5.4, B).

If the results are loaded successfully, the status indicator of the result bullet changes from gray to coloured.

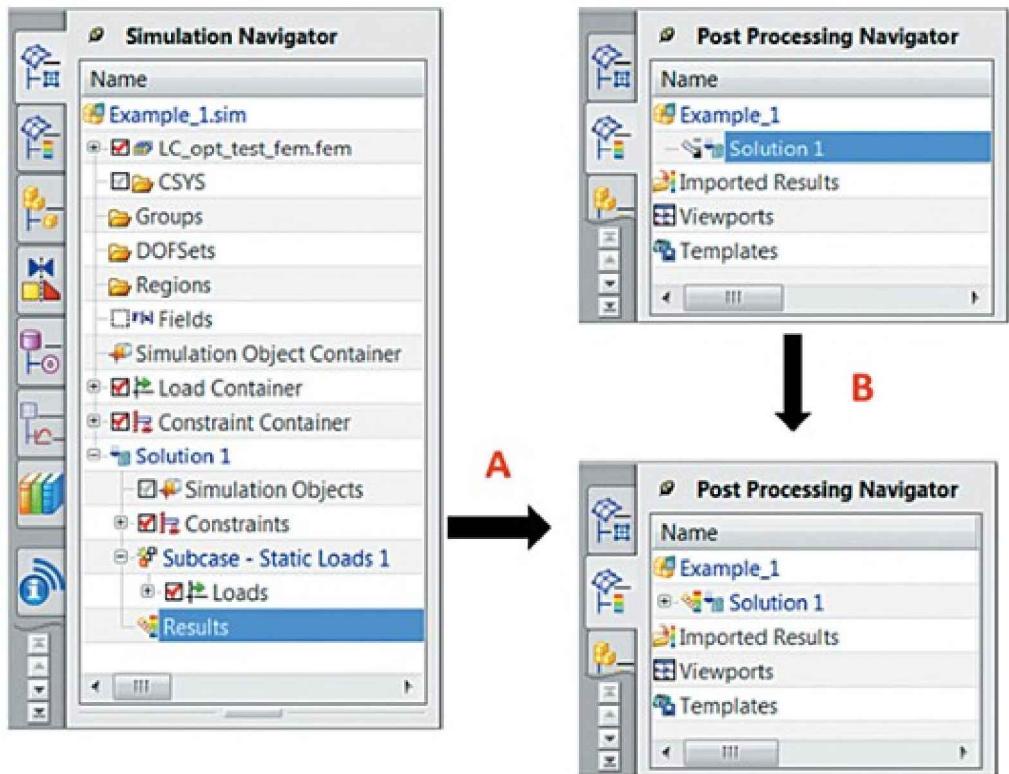


Figure 5.4. Loading simulation results into the postprocessor

The solution of a simulation problem comprises of scalar, vector, or tensor physical quantities. For example, temperature is a scalar quantity, displacement and reaction forces are vector quantities, while stresses and strains are second order tensor quantities. Vector quantities have three components, second order tensor quantities have three normal and three tangential components or three main components. In addition, for stress and strain tensors, there are derived invariant values: von Mises and Tresca equivalent values, octahedral, and mean values. For example, Figure 5.5 shows a result tree in the **Post Processing Navigator** tab that is typically encountered in deformable solid body mechanics problems.

There are three ways to represent results based on values computed in nodes:

- *Nodal*: one result value per node.
- *Elemental*: the result value is determined for the centroid of each element.
- *Element-Nodal*: a result value exists for each node of the element.

If a node is shared between multiple elements, that node has multiple result values. The element value at the centroid is calculated by taking an average of all the element's nodal results.

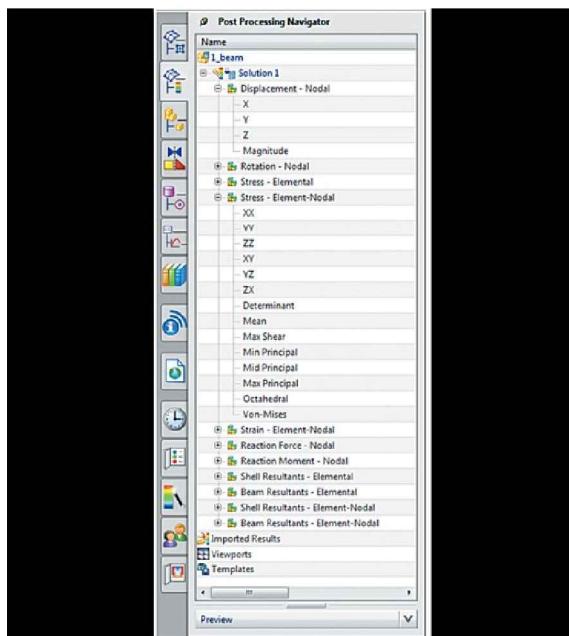


Figure 5.5. Results tree

Note that the type of results depends on the simulated problem. The following table lists the basic results types for deformable solid body mechanics problems.

Value	Value type
Offset	Vector
Rotation about the axis	Vector
Stresses	Symmetric second order tensor
Strains	Symmetric second order tensor
Strain energy	Scalar
Strain energy density	Scalar
Applied load	Scalar
Force	Vector
Moment	Vector
Temperature	Scalar
Heat flux	Vector

Temperature gradient	Vector
Contact forces	Vector
Contact pressure	Scalar
Safety factor	Scalar
Fatigue life	Scalar

5.1.2. Importing results files

You can import solution results created in NX Nastran or in third-party solvers. The following file formats are supported for importing results:

- *NX Nastran, MSC Nastran – (.op2)*
- *Structures P.E. – (.vdm)*
- *ANSYS – (.rst), (.rth)*
- *ABAQUS – (.odb), (.fil)*
- *I-DEAS – (.unv)*
- *I-DEAS – (.bun)*
- *LS-DYNA – (*.*)*

To load a results file, do the following:

- In the **Post Processing Navigator** tab, open the shortcut menu of the *Imported Results* node.
- In the **Import Results** dialog box, click *Browse*, select the appropriate file type, then select the relevant results file (Figure 5.6).

If the results file is successfully loaded into the tree, a new item appears in the **Post Processing Navigator** tab.

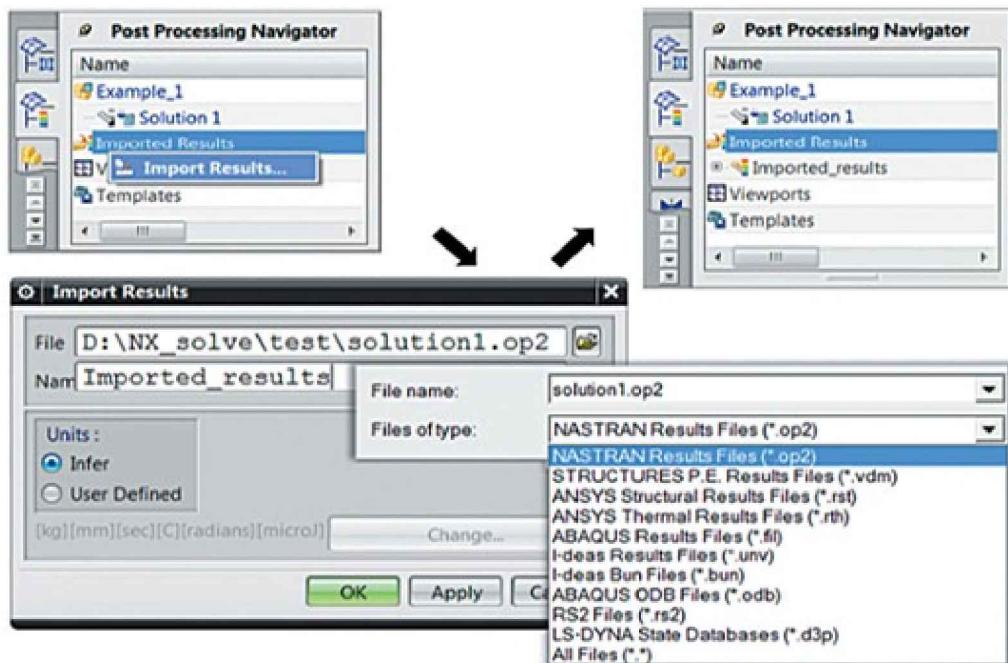


Figure 5.6. Importing a results file

Note that the layout of the results tree in the **Post Processing Navigator** tab also depends on the type of the solution. Here are some examples:

- Linear static analysis, solution type **SESTATIC 101** (Figure 5.7, A).
- Finding natural frequencies and modes, solution type **SEMODES 103** (Figure 5.7, B).
- Analysis of nonlinear processes variable in time, solution type **NLTRAN 129** (Figure 5.7, C).

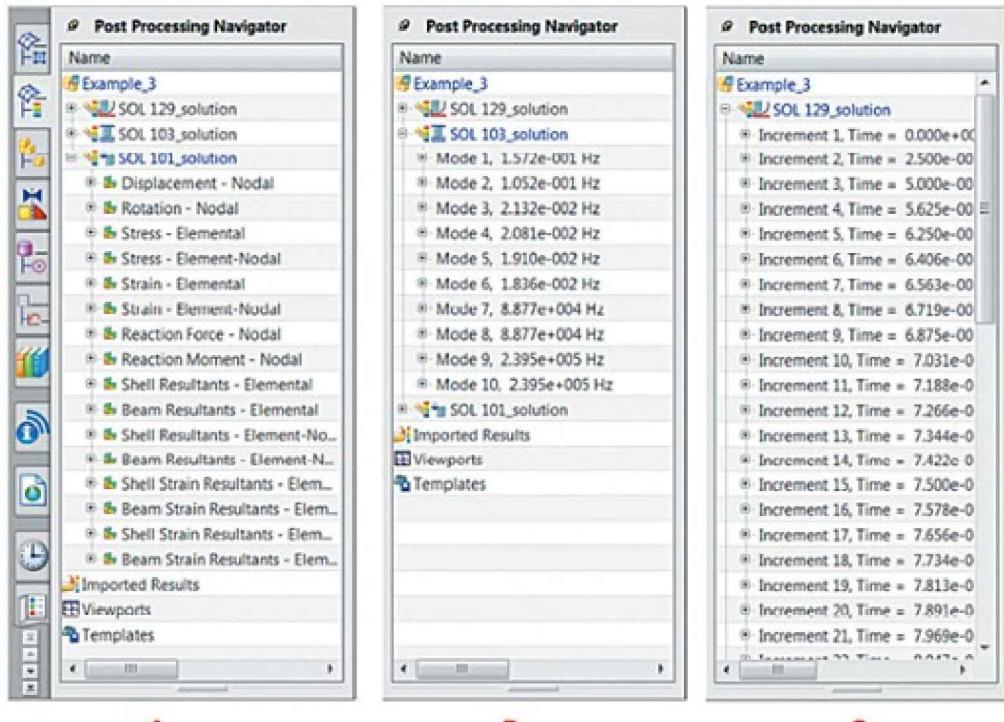


Figure 5.7. Results tree view

So, depending on the problem, the results tree in the **Post Processing Navigator** tab can contain results for various iterations or different modes with an identical set of result types.

The **Post Processing Navigator** tab displays information about the type of the solved problem next to the solution item, as well as next to the loaded imported results in the *Description* column (Figure 5.8):

- Name of the solver used
- Analysis type
- Name of solution type used

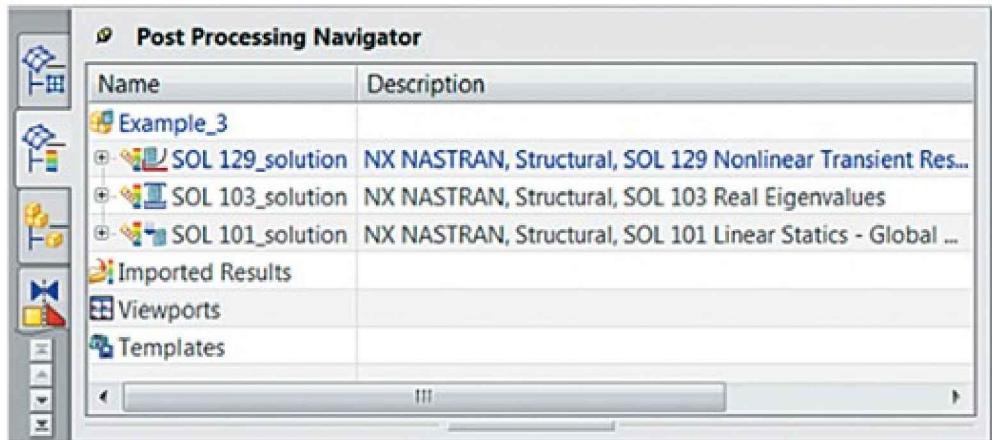


Figure 5.8. Description of results in the Post Processing Navigator tab

To unload simulation case results after processing and analysing them in the **Post Processing Navigator tab**, choose **Unload** in the shortcut menu of the current solution node. When you unload the results, the *op2* results file in the working directory is not deleted and remains available for reuse.

5.1.3. Selecting the value to display

Note that in the postprocessor, you can process only those values that were listed for output when the problem was formulated. These are the values written into the *op2* results file.

To display the results of interest, do the following:

- In the **Post Processing Navigator** tab, expand the results bullet (the name of the bullet is the same as the name of the solution). If the solution of the problem implies the existence of several result sets (unsteady processes, natural frequency and free vibration mode analyses), select the bullet corresponding to the required iteration (point in time) or mode.
- Select an item with the relevant results type and a component by double-clicking them.

In the example shown in Figure 5.9, displacement values along the X axes are selected for the iteration number two.

After you follow through this procedure, the **Post Processing Navigator** tab is updated as follows:

- Under the *Viewports* item, a *Fringe Plots* and a *Post View <n>* bullets appear (Figure 5.9).
- In the graphics area, a contour diagram is displayed (Figure 5.9).

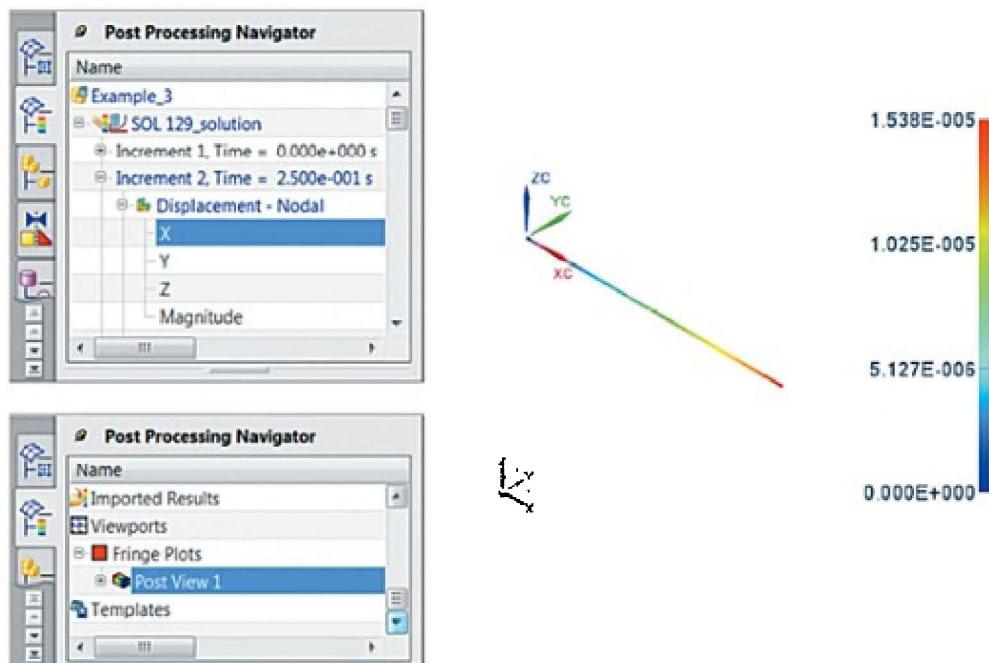


Figure 5.9. Displaying the X axis component of the displacement vector

In the column with the *Description* heading in the **Post Processing Navigator** tab, the following information is displayed next to each *Post View <n>* bullet (Figure 5.10):

- Displayed result.
- Name of the S/M file: name of the solution type.

Post View <n> contains bullets that correspond to the FE meshes of the model. The *Description* column of the **Post Processing Navigator** tab shows the following information for each FE mesh item (Figure 5.10):

- Number of elements
- Material ID
- Property ID

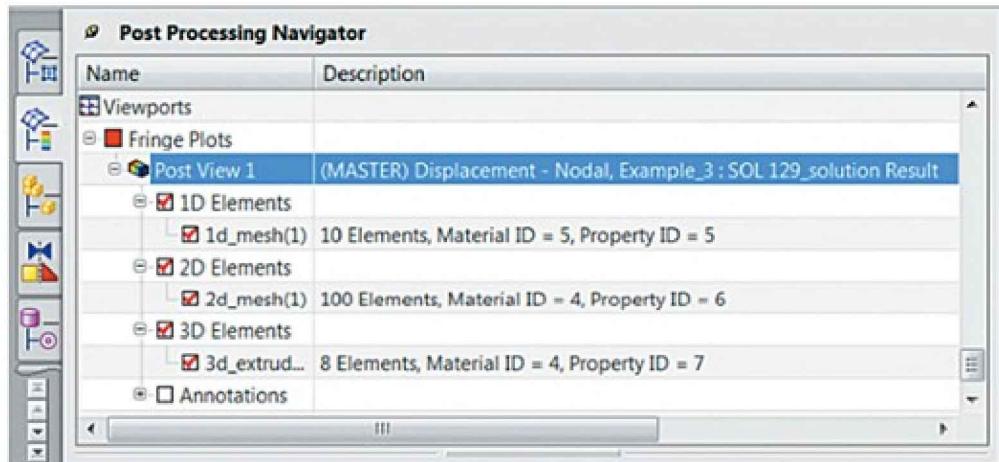
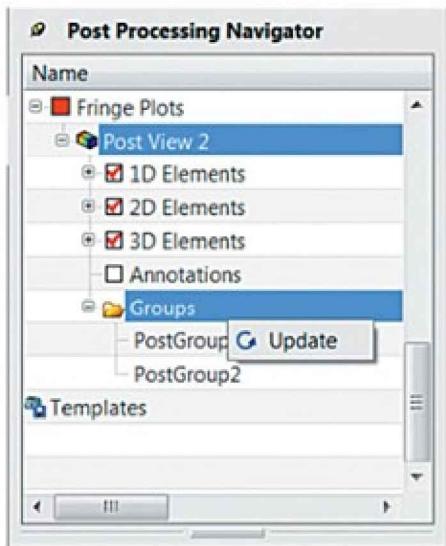


Figure 5.10. Post processing tree view for viewport selection

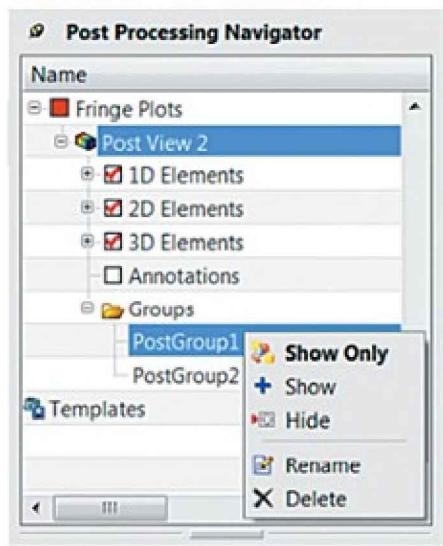
In addition, all groups of objects created in the **Simulation Navigator** tab are organized in separate *Groups* items in *Post View <n>*. If the groups are created after loading the results, in the shortcut menu of the *Groups* bullet, choose **Update** (Figure 5.11, A).

You can use the following shortcut menu commands for the groups themselves (Figure 5.11, B):

- Choose **Show Only** to show only objects belonging to the group.
- Choose **Show** to add objects of this group to displayed elements.
- Choose **Hide** to hide objects belonging to the group.
- Choose **Rename** to enter a new name for the selected group.
- Choose **Delete** to delete the group.



A



B

Figure 5.11. Shortcut menu of the Post View items

A. Shortcut menu of the Groups item. B. Shortcut menu of the object item under the Groups item

You can manage the display of results for a particular FE mesh or group in the graphics area by selecting or clearing the check box next to the corresponding bullet.

You can display several results in the graphics area at the same time. To do so, display one of the results then choose **Overload** in the shortcut menu of a different result. For example in Figure 5.12, the graphics area shows results for the second and the last iterations simultaneously. In the active *Fringe Plots* bullet with *Post View <n>* sub-bullet, an additional *Post View <m>* sub-bullet appears. The active *Post View* is marked with a filled red circle and a *MASTER* label in the *Description* column. When you switch from *Post View <n>* to *Post View <m>*, you will see the palette and heading information change in the corresponding display zone (Figure 5.12, A, B).

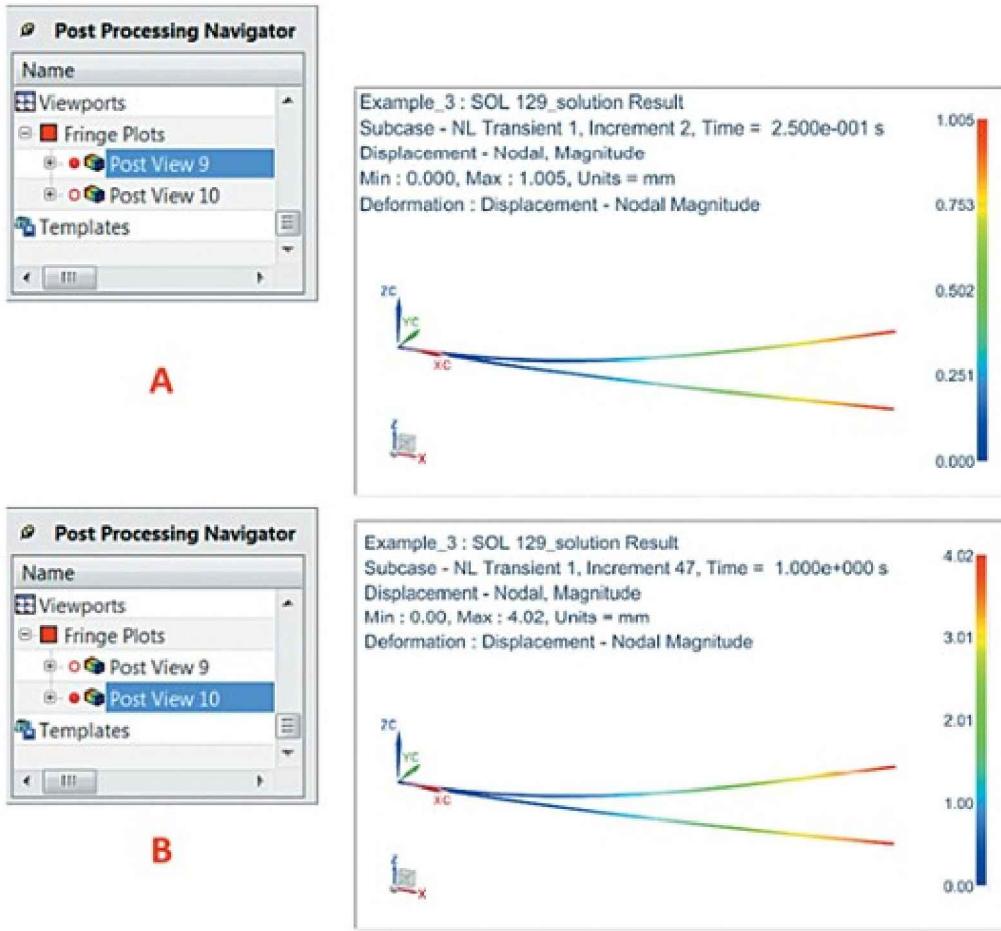


Figure 5.12. Using the Overload command

- A. The display corresponds to the displacement at the second iteration. B. The display corresponds to the displacement at the forty-seventh iteration

This tool is convenient because you can display different results for different groups or different FE sets simultaneously. You can select and clear check boxes next to element sets and use object set operations to manage the visualization of results. Figure 5.13 shows an overlay of two Post Views: for the 3D object, *Displacement* results (*Post View 13*) are shown, while for the shell, *Rotation* results are show (*Post View 12*).

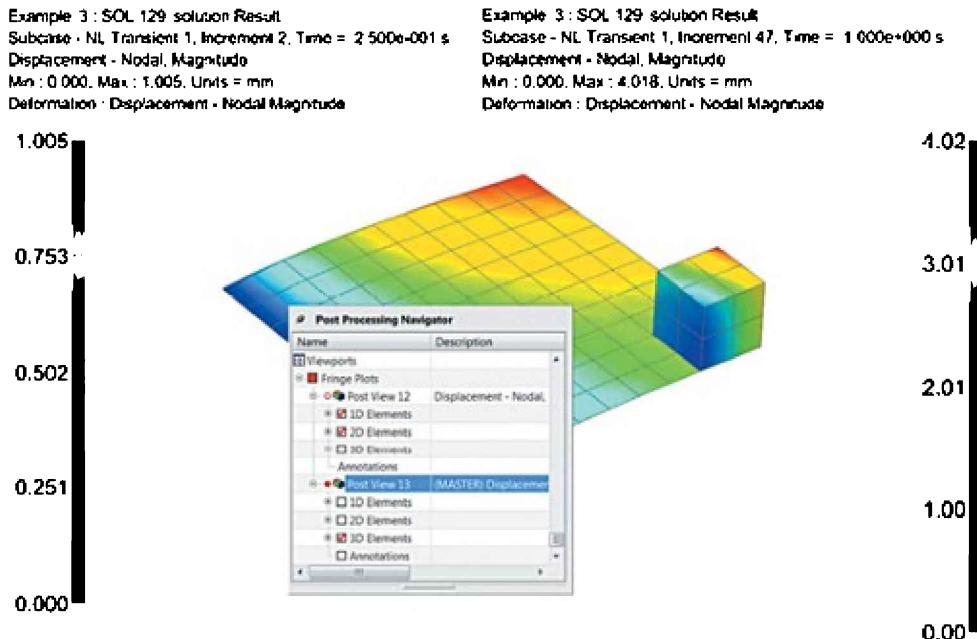


Figure 5.13. Overlaying two Post View n

5.1.4. The Layout Manager toolbar

For your convenience, **NX Advanced Simulation** supports dividing the graphics area into several zones. You can use up to nine zones to simultaneously display several views of the object as well as animations, plots and so on.

You can create several zones in the graphics area using commands of the **Layout Manager** toolbar: *Single View*, *Side by Side*, *Top and Bottom*, *Four Views*, *Nine Views* (Figure 5.14).

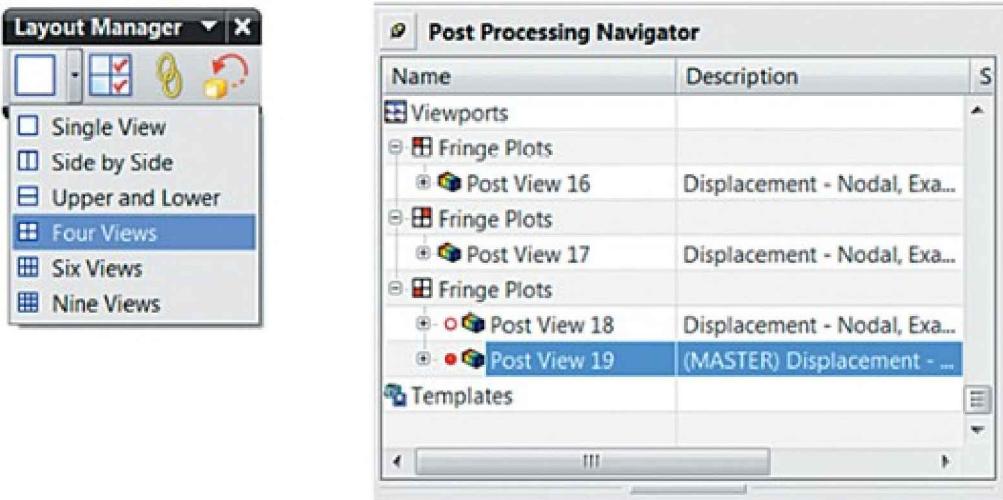


Figure 5.14. View layouts

You can choose the results shown in each particular zone (Figure 5.15).

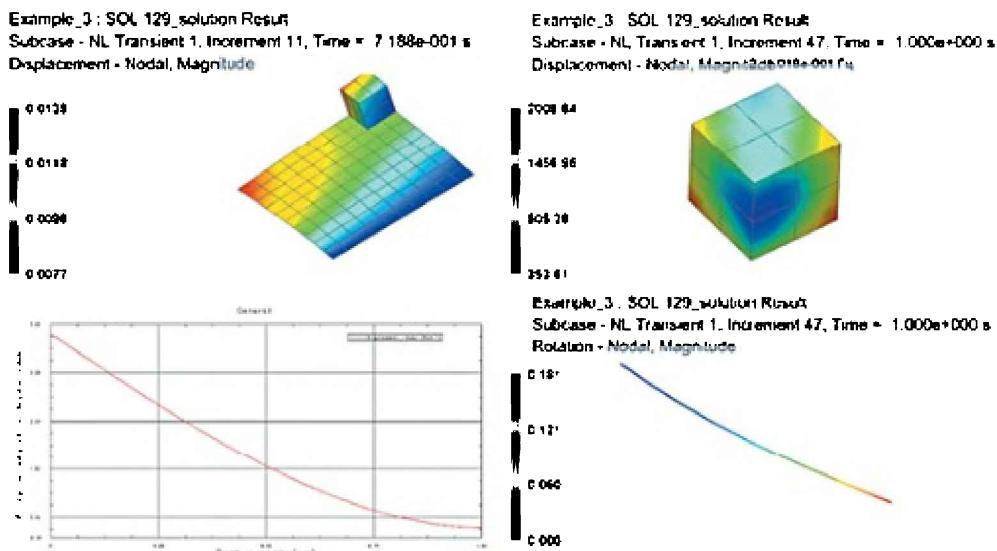


Figure 5.15. Four-view layout of the graphics area

When you specify a viewport for results in the *Viewports* bullet, the corresponding *Fringe Plots* bullets are created with *Post View <n>* subbullets. The indicator of the current state of *Fringe Plots* bullets is synchronized with the specified zone arrangement in the graphics area. The filled cell of the *Fringe Plots* bullet indicator corresponds to the zone where the *Post View <n>* is displayed (Figure 5.13).

The Post View <n> bullet with “MASTER” in the *Description* column corresponds to the active graphics area.

The **Layout Settings** command is relevant only when the graphics area window is divided into several zones. Click to open the **Viewport Settings** dialog box with the following **Viewport Settings** (Figure 5.16):

- Select *Active in Work Viewport* to work only in the active graphical display zone by default.
- Select *User Selected Viewport* to select a custom results display zone. To make a selection, right-click the results bullet in the tree and choose **Plot**, then select the results display zone you need.
- Select *User Specified Viewport* to automatically determine the zone change sequence (the sequence for each layout is specified in the boxes that appear when you select this option).

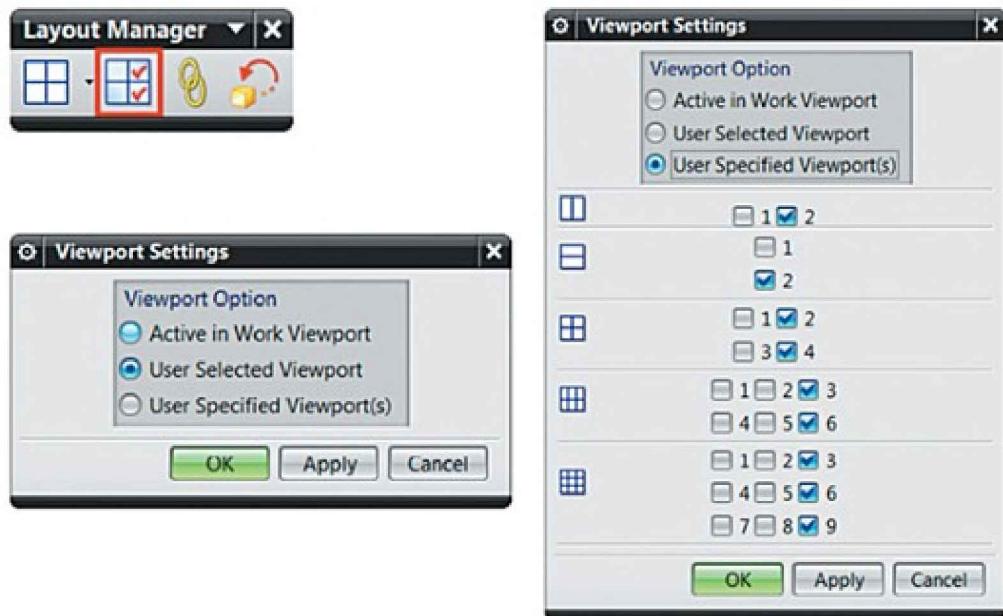


Figure 5.16. *Viewport Settings* dialog box

Consider the **View Synchronize** command. This command is only relevant when the graphics area is divided into several zones. When you use this tool, the **Viewport Settings** dialog box opens, and you can select zones for synchronizing. Select the necessary zones and click **OK**. When you rotate or move the model in one of the selected zones, models in all the other linked zones will rotate and move in the same way.

When you have finished viewing the results, use the ***Return to Model*** command to bring back the geometry or finite-element representation of the model in the graphics area. If the graphics area is divided into zones, you need to select the zone in which to cancel the results display. You can perform a similar action by deleting the corresponding *Post View <n>* bullet in the **Post Processing Navigator** tab.

5.2. Basic display features

The primary tool for setting up the results display in the graphics area is the ***Edit Post View*** command of the **Post Processing** toolbar. You can also open the dialog box of this command through the shortcut menu of a *Post View <n>* bullet by choosing ***Edit***. Note that the command is activated only when a result is displayed in the graphics area.

The ***Post View*** dialog box has four tabs: *Display*, *Legend*, *Edges&Faces*, *Annotation*. The following section describes the most important options of each tab.

5.2.1. Setting up the results display

The most common method of displaying results for FE modeling is the contour diagram display. NX Advanced Simulation postprocessor implements the following display types of results contour diagrams (Figure 5.17):

- *Smooth*: smooth display of results (Figure 5.18, A).
- *Bounded*: display with sharp boundaries (Figure 5.18, B).
- *Element*: displaying a value for each element (Figure 5.18, C).
- *Iso-Line* (Figure 5.18, D).
- *Iso-Surface* (Figure 5.18, E).
- *Cubes* (Figure 5.18, F).
- *Spheres* (Figure 5.18, G).
- *Arrows*.
- *Tensors* – (Figure 5.18, H, I).
- *Streamlines*.

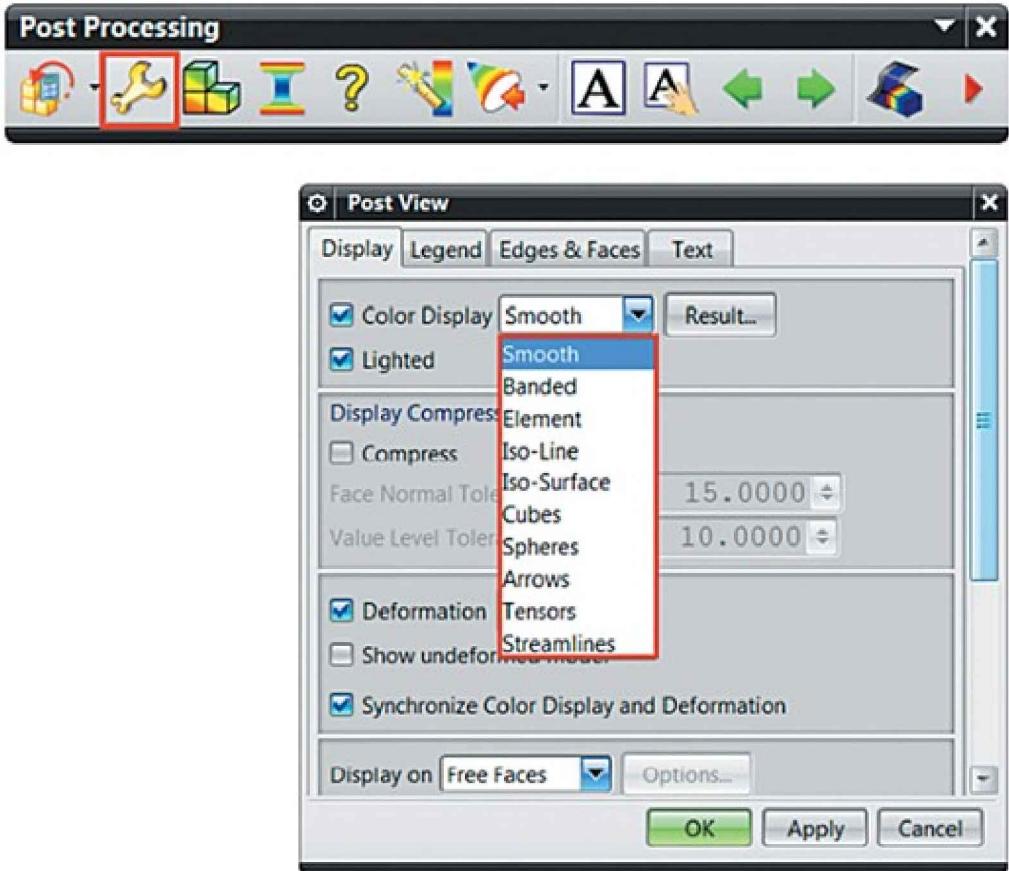


Figure 5.17. Changing the results display type

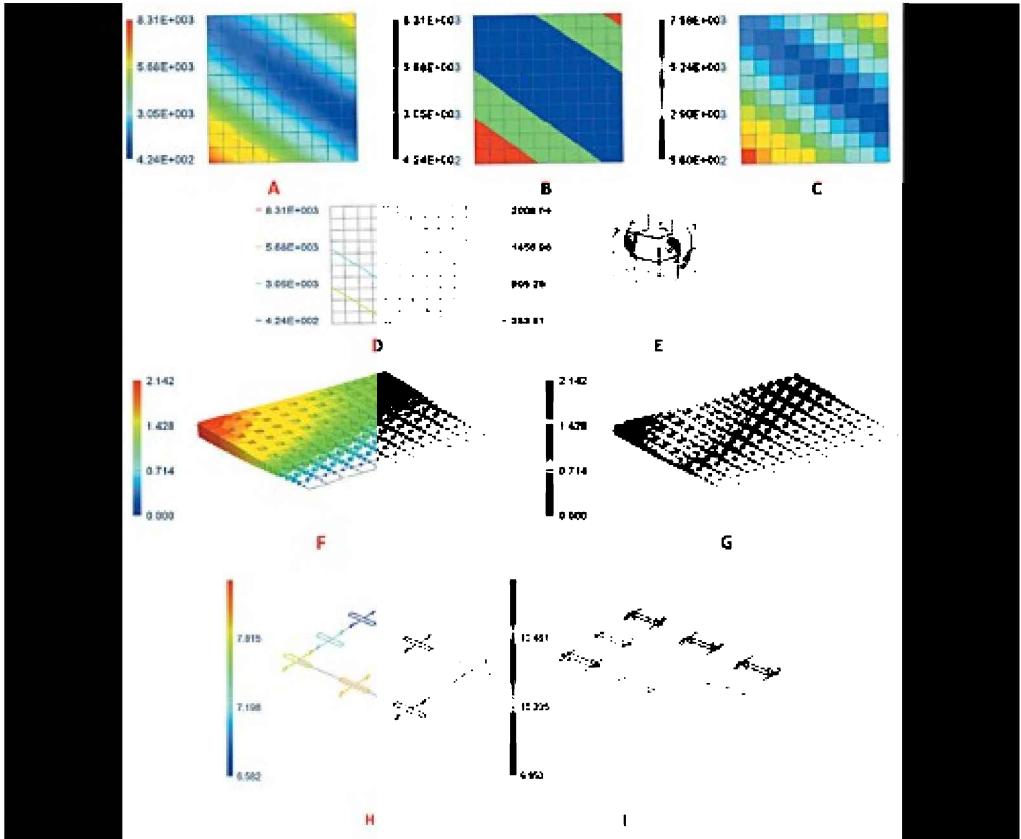


Figure 5.18. Different results display types

You can select a value and the corresponding result component for display in the **Smooth Plot** dialog box. You can access this dialog box either using the **Set Result** command of the **Post Processing** toolbar or by clicking **Results** next to the drop-down list of contour diagram display types of the **Display** tab in the **Postprocessor View** dialog box. Depending on the selected display type (Figure 5.17), the displayed results selection dialog box can have different titles: **Smooth – Smooth Plot**, **Bounded – Bounded Plot**, **Element – Element Plot**, and so on (Figure 5.19).

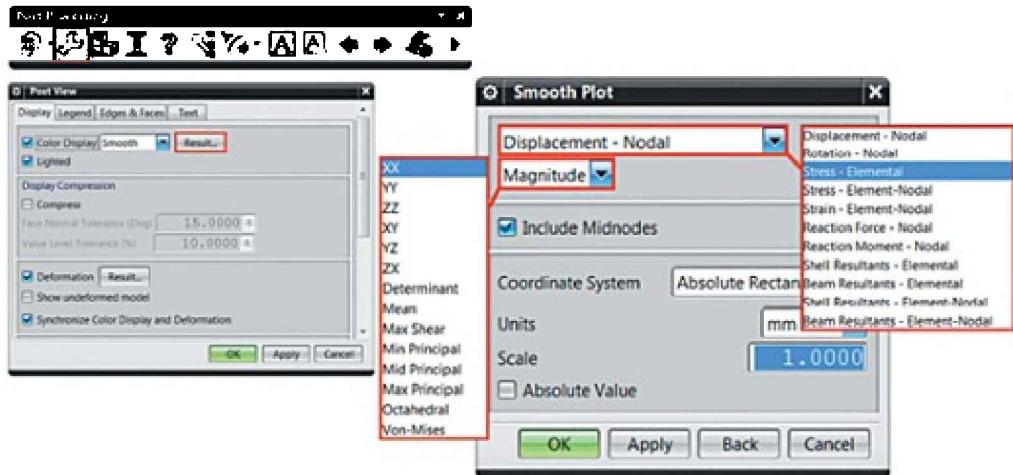


Figure 5.19. Selecting results for display

Note that the layout of the displayed results selection dialog box depends on the contents of the results tree.

Consider some of the possible options:

- When you solve natural frequency and mode problems, all produced results are listed in the first drop-down list of the displayed results selection dialog box. Each mode is represented by a name with the pattern *Mode <m>* containing a sequential number and a numerical value of natural frequency (Figure 5.20, A).
- For problems solved in several iterations, you can select *Increment <m>* with the number of the iteration of interest (Figure 5.20, B).
- You can also select results of a simulation case (Figure 5.20, C).

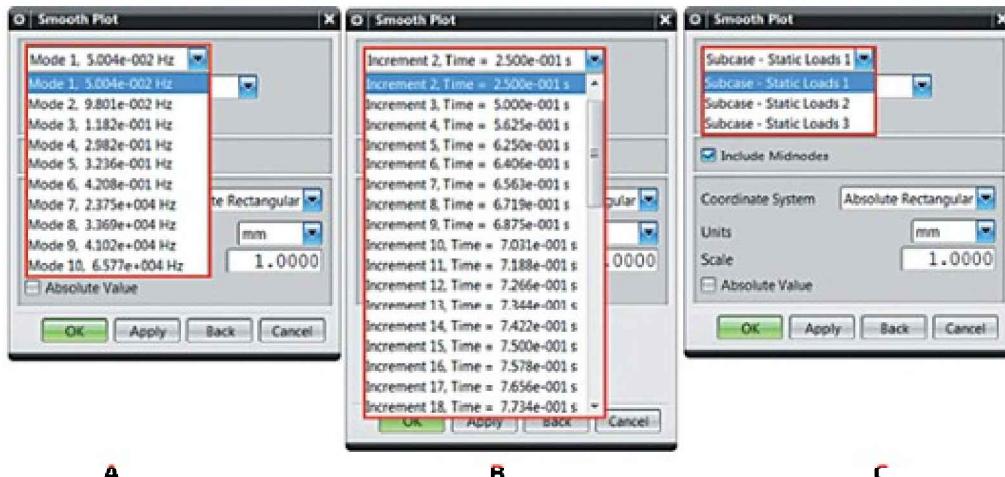


Figure 5.20. Displayed results selection dialog box

When you select *Smooth*, *Bounded*, *Iso-Line*, *Iso-Surface* display types, you can ignore values produced for midnodes. Simply clear the *Include Midnodes* check box.

If you select *Cubes*, *Spheres*, *Tensors* display types, additional boxes appear in the dialog box for selection of displayed results:

- You can use the *Draw Marks at* box to select the location of result display objects (cubes, spheres, tensors). The following locations are available: *Elements*, *Nodes*, (Figure 5.21, A, B).
- You can use the *Hide Marks Below* box to hide all results below the specified value (Figure 5.22).

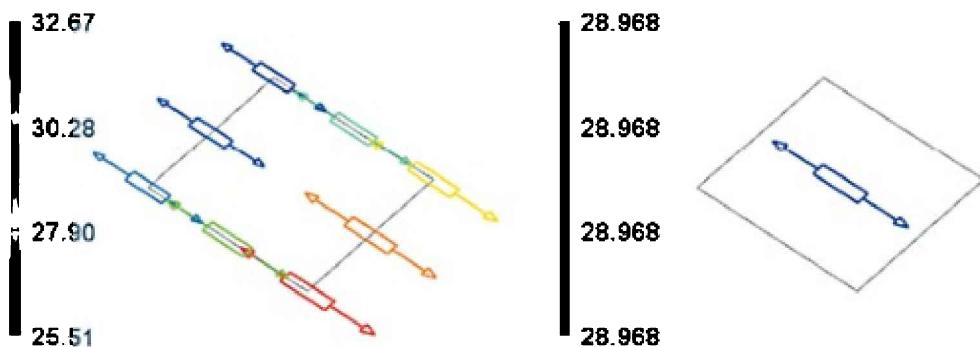


Figure 5.21. Displaying results for a single element with a tensor.
A. At nodes. B. At an element

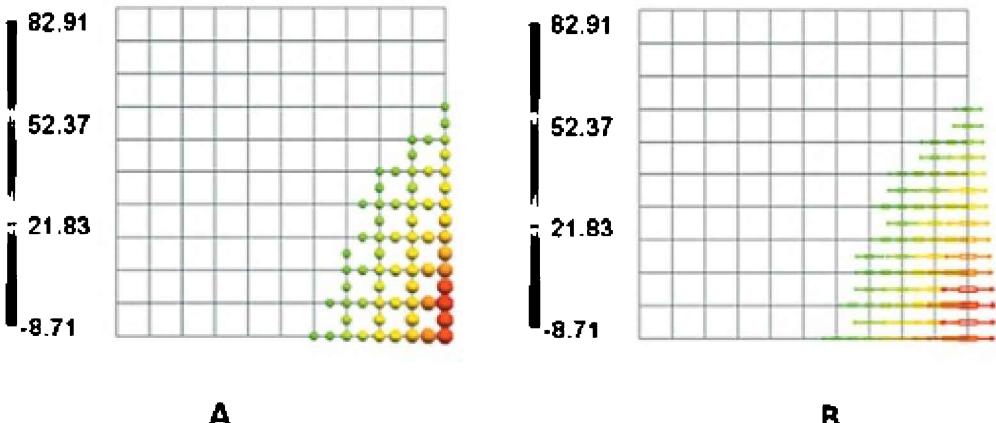


Figure 5.22. Displaying results using the hide marks below 50% option A. Spheres. B. Tensors

When you solve problems using beam elements, to display *Nodal* and *Element – Nodal* results, you can select the section point (Figure 5.23). To do so, in the displayed results selection dialog box (*Smooth Plot*, *Banded Plot*, and so on), in the *Beam* drop-down list, select the required *Recovery Point*. The results for beam structures can be displayed at four specified section points: *C*, *D*, *E*, *F*. You can also display the maximum or minimum value out of the four for each section.

Use the *Calculate Beam Results from Forces and Beam Geometry* option for 1D beam elements based on nodal forces to determine stress and strain when you introduce additional blends into the section.

If you need to solve problems that involve shell 2D elements, select the shell side *Bottom* or *Top* to display results (stress or strain). For more details, refer to para. 5.3.2. You can also display the following result combinations for shells: *Maximum*, *Minimum*, *Average/Middle*, *Top and Bottom*, *Bending* (Figure 5.23).

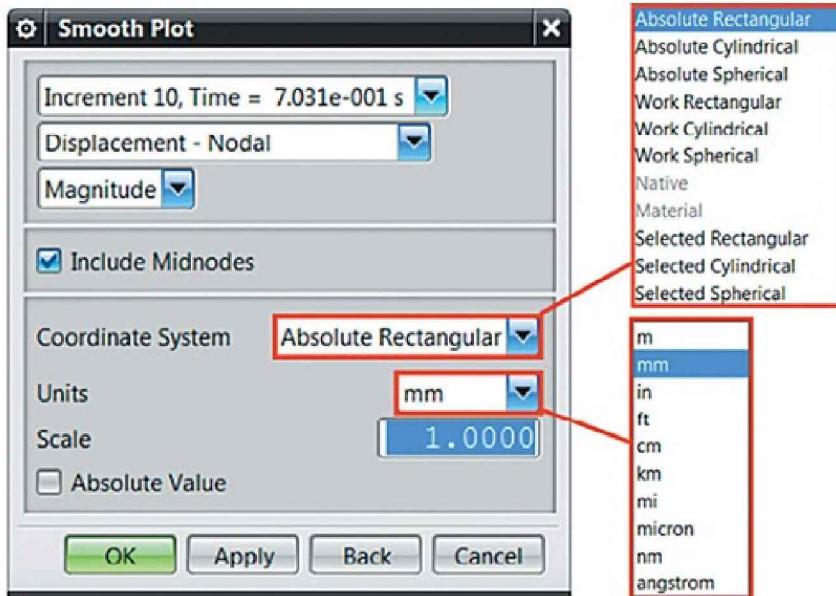


Figure 5.23. Setting up the results display for shell and beam elements

When using the *Arrows* results display type, ***Smooth Plot***, ***Banded Plot*** and other dialog boxes for selecting displayed results can simultaneously show several components with corresponding numerical values. To do so, select the relevant check boxes (Figure 5.24).

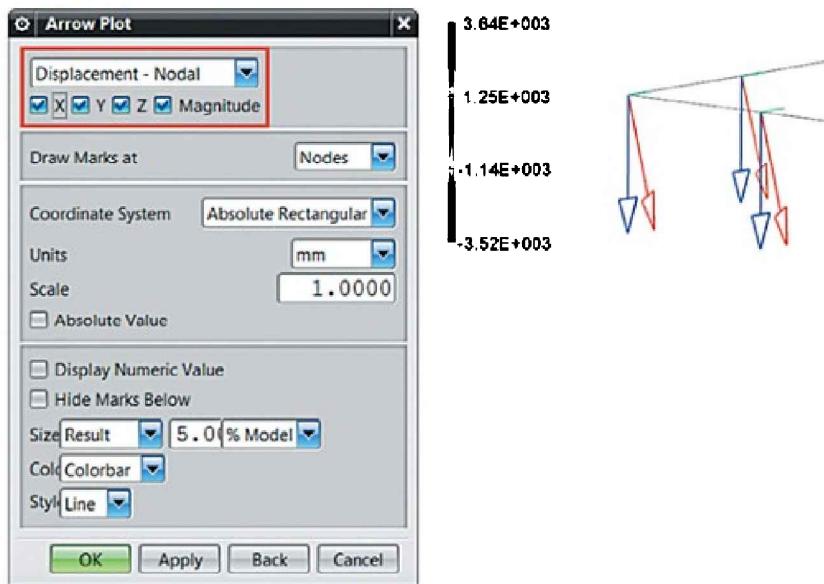


Figure 5.24. Displaying all components of the displacement vector and resultant vector

In the displayed results selection dialog box, you can select the *Coordinate System* to use for displaying results (Figure 5.25).

Moreover, you can present calculation results in different units. To change the unit, simply edit the *Units* option. To display scaled results, enter a scaling factor in the *Scale* box. Use the *Absolute Value* option to display the absolute values of the selected variable.

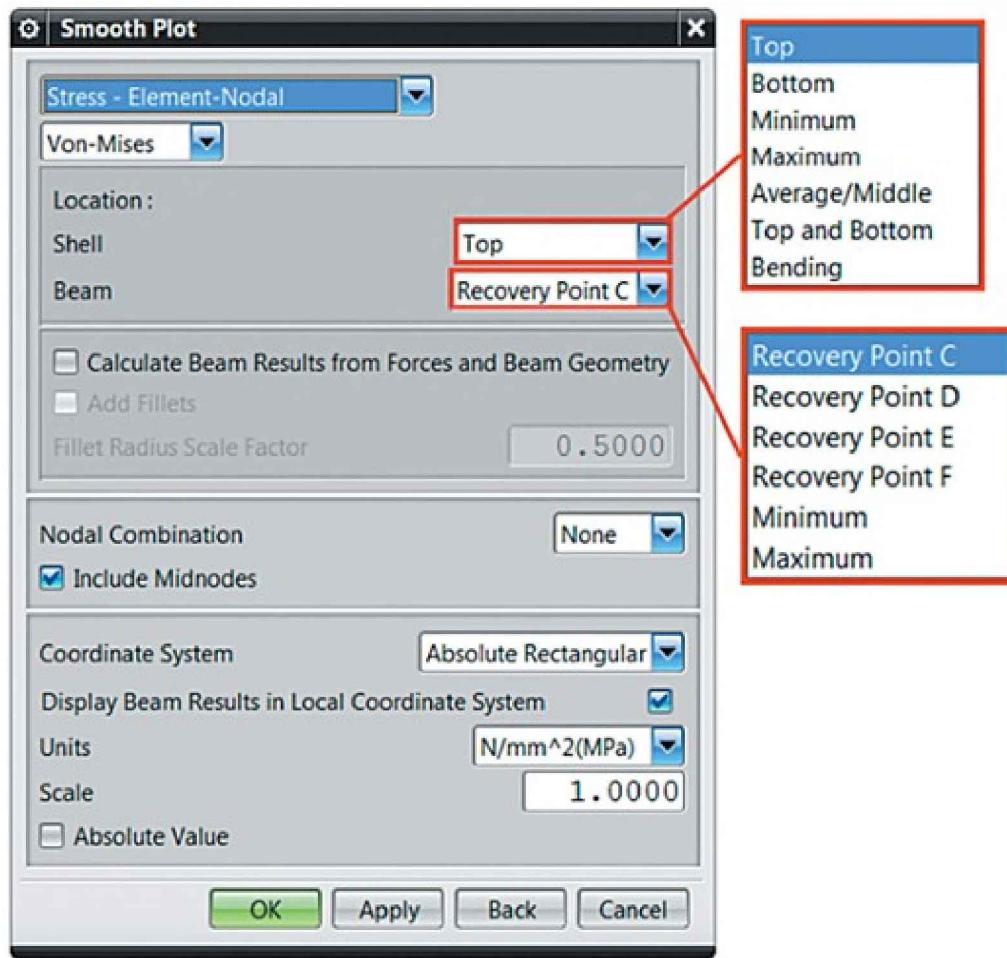


Figure 5.25. Setting up results display options (coordinate systems, scale, recovery points, and units of measurement)

Here is a description of display settings in the *Display* tab in the *Post View* dialog box.

You can display the deformed state of the simulated structure by selecting the *Deformation* check box in the *Display* tab. To open the *Deformation*

dialog box and set up display options for the deformed state, click *Results*. You can make the following settings:

- Select variables to use for rendering the deformed state: *Displacement – Nodal, Rotation – Nodal, Reaction Force – Nodal, Reaction Moment – Nodal*.
- Select a component of the value that corresponds to a coordinate system axis: *X, Y, Z*, or its absolute value.
- Enter the *Scale* value to specify the display scale factor.
- Select the scaling method: *%Model* or *Absolute*.
- Select the *Reference Node*, that is, the node relative to which the scaling operation is performed.

If you select the *Deformation* option, you can also display the deformed and undeformed views of the model simultaneously. To do so, select the *Show undeformed model* check box. The undeformed state is displayed using transparency.

If you retain default settings, the results are displayed on free faces of elements in accordance with the *Free faces* option in the *Display on* list. You can also display results for the whole volume using the *Volume* option, or depending on the position of the specified cutting plane using the *Cutting Plane* option in the *Display on* list. Displaying results on a cross section using the *Cutting Plane* option, which allows viewing results inside the model, needs to be described in greater detail. If you select this option, the *Options* command becomes available, which you can use to open the ***Cutting Plane*** dialog box with the following settings for positioning and visualizing the cutting plane (Figure 5.26):

- In the *Cut Plane* list, select a coordinate system and axes to create a section.
- You can position the cutting plane using a slider or by entering a numerical value in the relevant box. Select the *Automatic Update* option to modify the position of the cutting plane automatically and synchronously with the movement of the slider without clicking *Apply*.
- In the *Clip Side* list, you can define the location of the cut-off part relative to the direction of the selected coordinate system parameter: *Positive* corresponds to the part behind the cutting

plane (Figure 5.27, A), *Negative* corresponds to the part in front of the cutting plane (Figure 5.27, B), while *Both* corresponds to displaying results only on the cutting plane itself (Figure 5.27, C).

If the *Show Feature Edges* and *Show Clipped Ghost* check boxes are selected, edges and faces of the cut-off part are shown in grey.

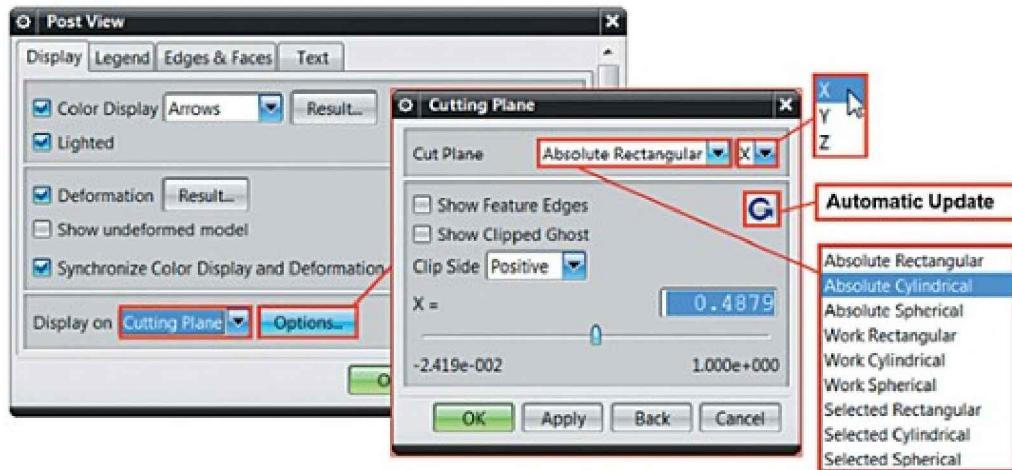
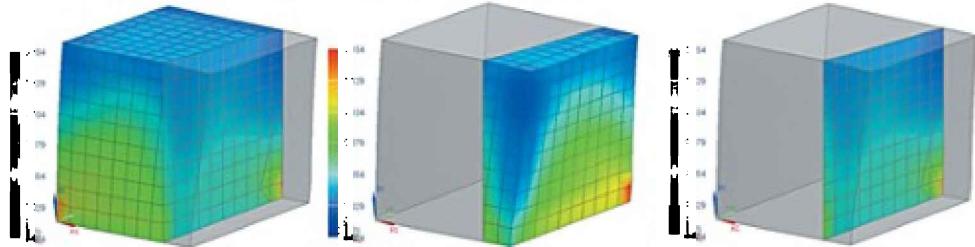


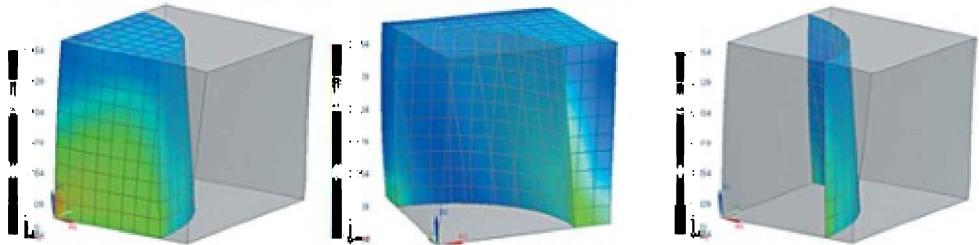
Figure 5.26. Setting up the cutting plane position

For example, Figure 5.27 shows the results display depending on the *Cut Plane* and *Clip Side* options.

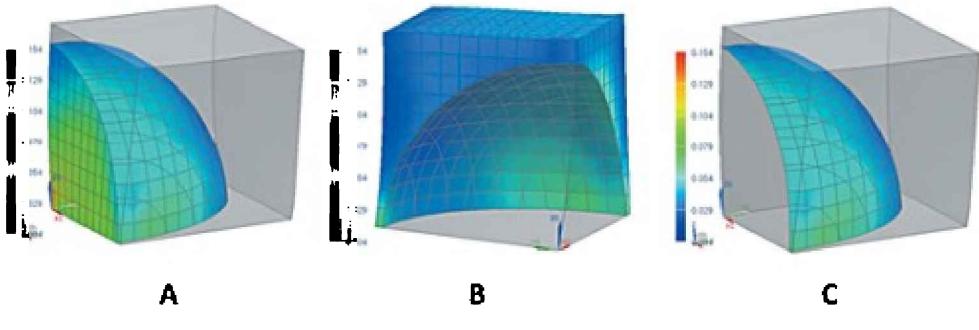
Coordinate System – Absolute Rectangular (parameter - OX)



Coordinate System – Absolute Cylindrical (parameter - Radius)



Coordinate System – Absolute Spherical (parameter - Radius)



A **B** **C**

Figure 5.27. Displaying results using a cutting plane.

A. Clip side Positive. B. Clip side Negative. C. Both.

When you solve axisymmetric problems, in the **Post View** dialog box, in the *Display on* list you can select an extra **3D Axi-symmetric** option that you can use to create high-quality realistic 3D display for axisymmetric results (Figure 5.28).

You can set up the 3D axisymmetric display in the **3D Axi-symmetric** dialog box:

- Enter the *Revolve Angle* to specify the sector angle value for the displayed results.
- Enter the *Number of sections* to specify the number of radial sections to manage the quality of displayed image.

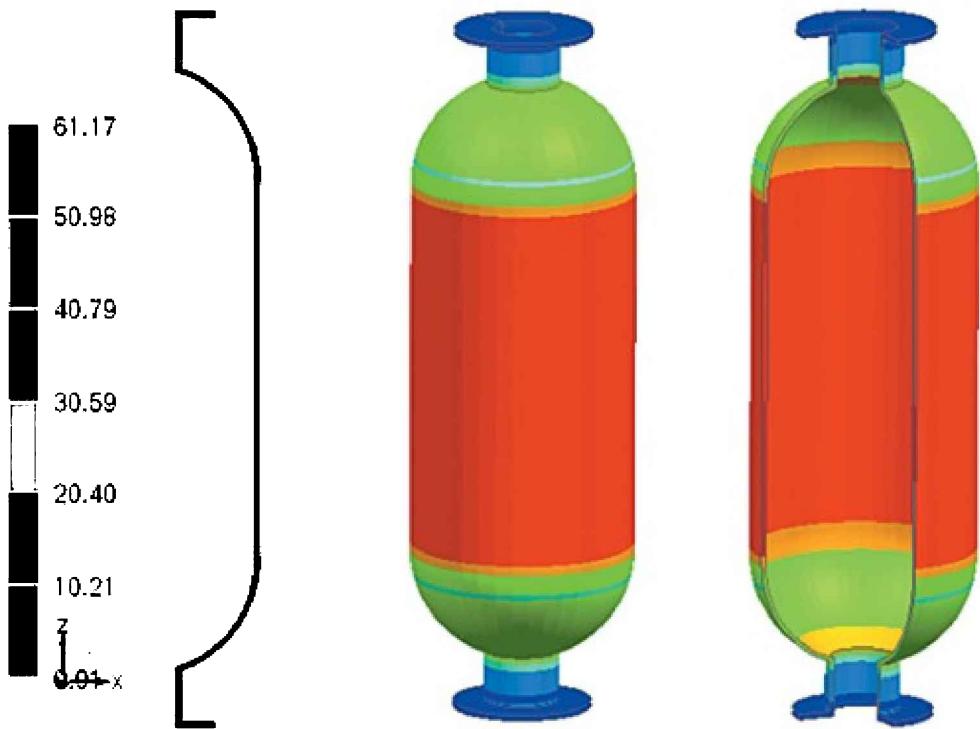


Figure 5.28. 3D axisymmetric display

5.2.2. Graphics area display settings

When you display simulation results in the graphics area, in addition to the contour diagram itself you can see a colour scale palette and a header. The header together with the palette is customarily referred to as the legend of the current result.

The legend text includes the following information: the current simulation model file, the solution, the step and simulation case, the name of the value currently displayed and its component, result output parameters, the coordinate system, maximum and minimum values, and so on (Figure 5.29).

The palette is a colour scale that correlates colours with numerical values.

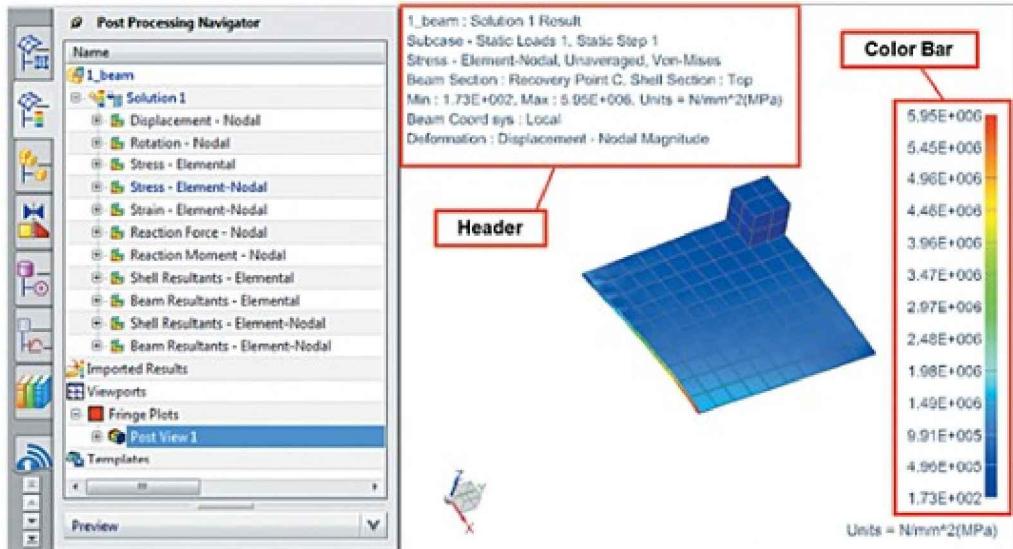


Figure 5.29. The result legend

You can manage the legend display in the *Legend* tab of the *Post View* dialog box (Figure 5.30).

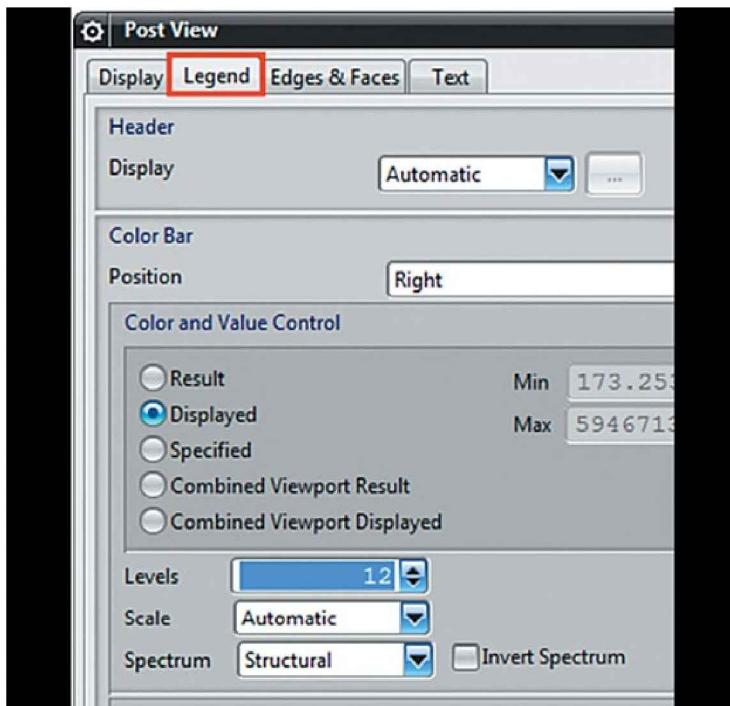


Figure 5.30. Setting up the legend options

There are two ways to display the header:

- Select *Automatic* to display default information.
- Select *Customized* to set up the content of the header (specify the number of lines in the header and the content of each line) (Figure 5.31).

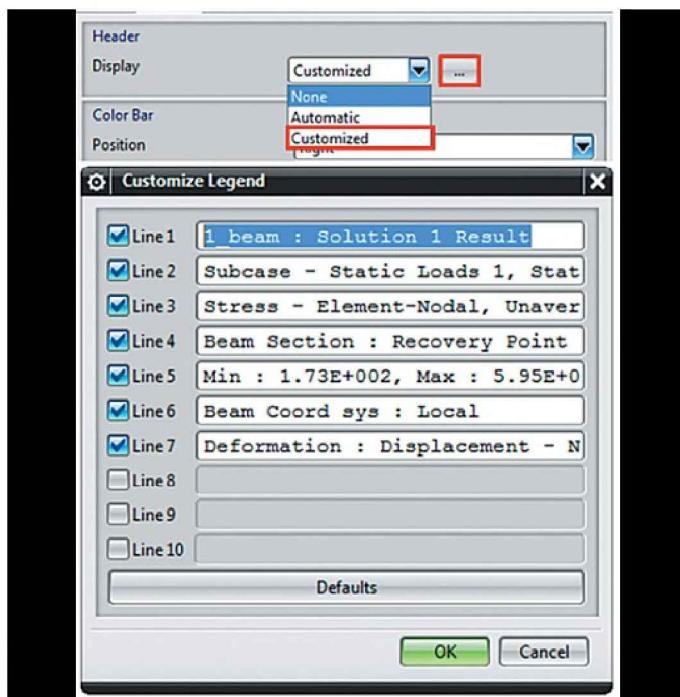


Figure 5.31. Setting up the header display

There are three ways to position the palette in the graphics area: you can show it on the right, on the left, or hide it altogether.

In the *Color and Value Control* group, you can specify:

- The minimum, maximum, and threshold values of the palette.
- *Levels*, that is, the number of sections in the palette.
- The *Scale* that determines the method used to set the range of values for each colour: *Automatic*, *Linear*, *Logarithmic*. You can also specify value ranges for each colour manually by selecting the *Custom Overwrite* check box.
- The *Spectrum* option that defines the palette type (Figure 5.32): *Structural* for deformable solid body mechanics problems, *Thermal* for thermal problems, *Grayscale* for display in shades of gray, *Stoplight* for three-color grading (the colours of traffic lights: green,

amber, red).

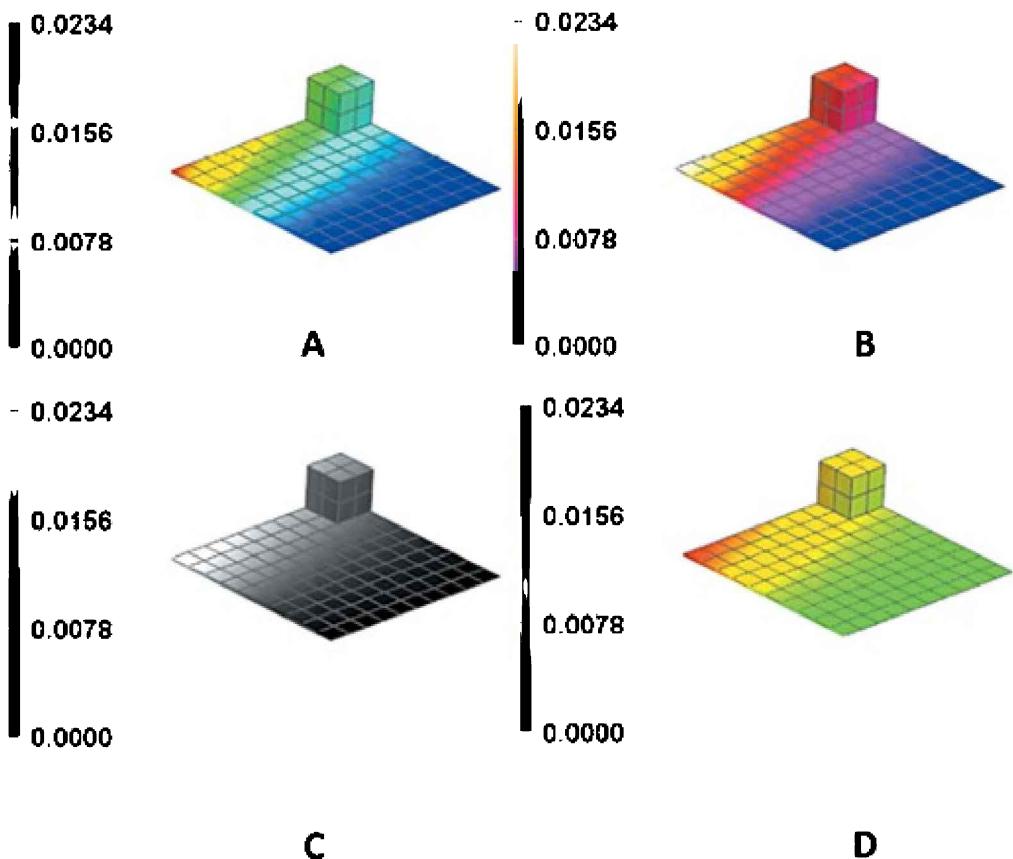


Figure 5.32. Palette spectra.

A. Structural. B. Thermal. C. Grayscale. D. Stoplight

Note that when you divide the graphics area into several zones while displaying several results, you can define a common scale for all results (Figure 5.33). To do so, in **Post Processing Navigator**, select all *Post View <n>* items of interest by holding the *Ctrl* key and clicking them. After that, in the **Post View** dialog box, under *Color and Value Control*, select *Specified* and use the *Get min/max for all selected postviews* command.

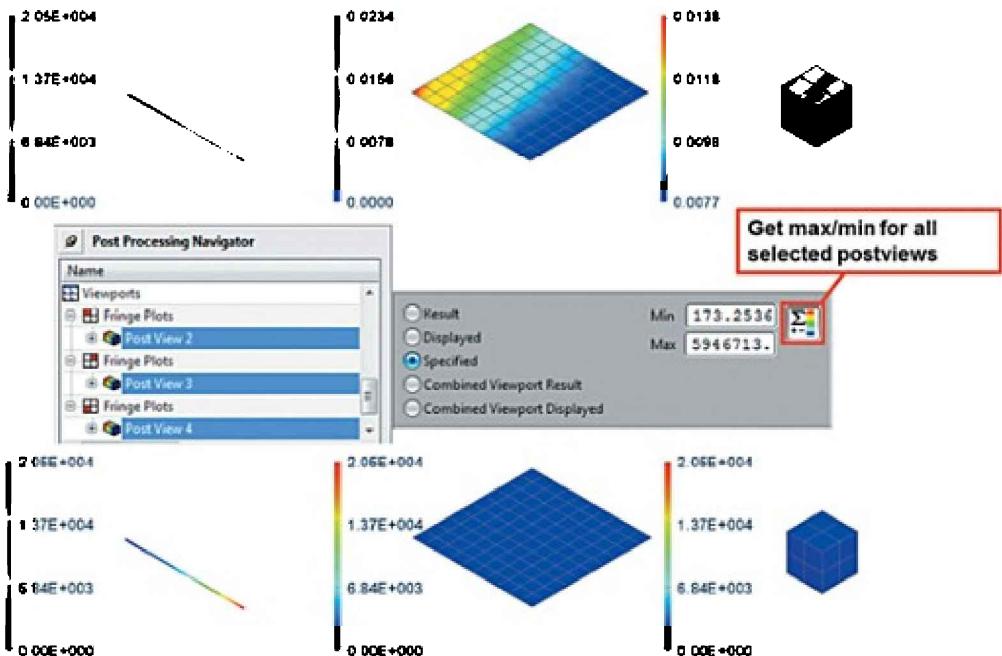


Figure 5.33. Common scale for several results

You can also select the areas of the model where result values are greater than the maximum value or less than the minimum value specified for the palette. You can set up the display of areas where results overflow the specified maximum value for the given range using the *Overflow* option. To set up the display in the reverse case where the result value is less than the minimum, use the *Underflow* option. You can select the following display methods:

- Select *Shared* to specify a color for the relevant areas (Figure 5.34, B).
- Select *Translucent* to display the areas with transparency (Figure 5.34, C).
- Select *Clipped* to hide the areas altogether (Figure 5.34, D).

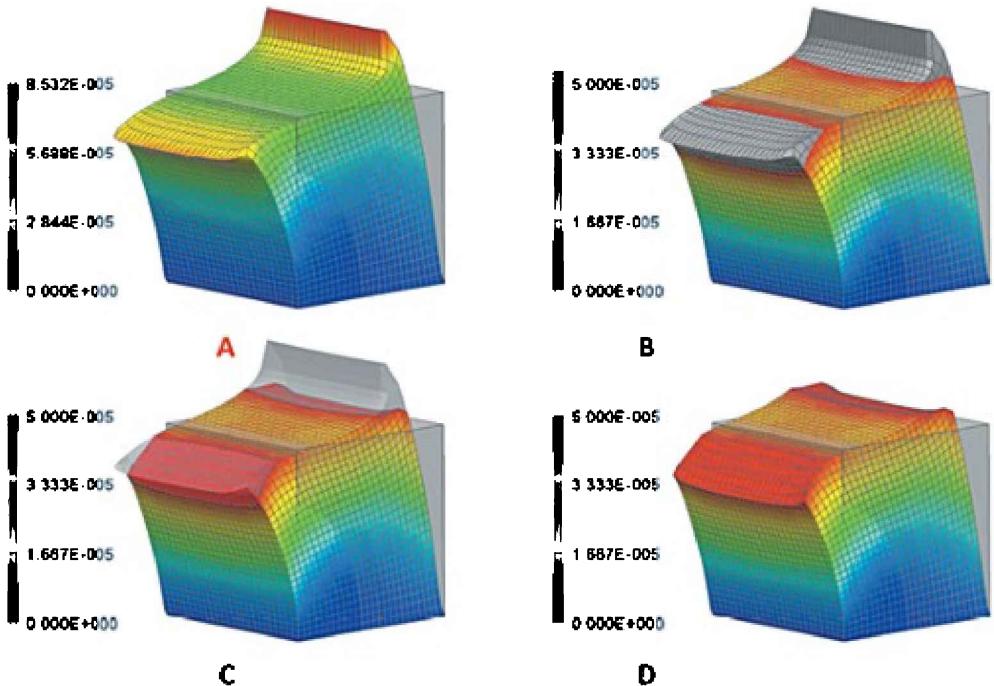


Figure 5.34. Displaying elements whose results fall outside the palette range
A. None. B. Shared. C. Translucent. D. Clipped

5.2.3. Setting up edge and face display

You can find additional display options for edges and faces of finite elements and geometry in the *Edges&Faces* tab (Figure 5.35).

The *Edges* list in the *Primary Display* group contains the following options:

- Select *External* to display only edges of external finite elements.
- Select *Feature* to display only edges of the structure's geometry.
- Select *Wireframe* to display edges of all elements.
- Select *None* to hide all geometry and element edges.
- In the *Faces* list you can select transparent or opaque display of faces (Figure 5.36).

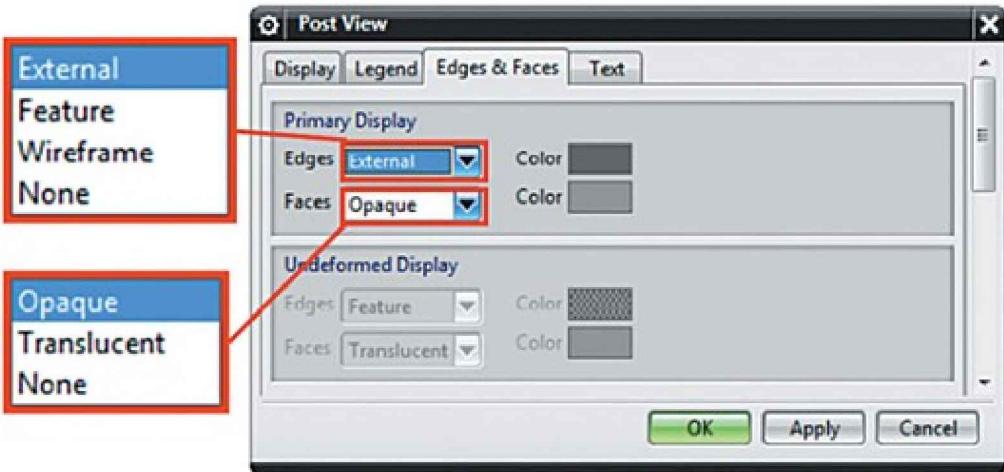


Figure 5.35. Setting up display of edges and faces of elements and geometry

When you display an undeformed state of the model, you can additionally set up options of the *Undeformed Display* group, which are similar to edge and face visualization settings of the deformed display.

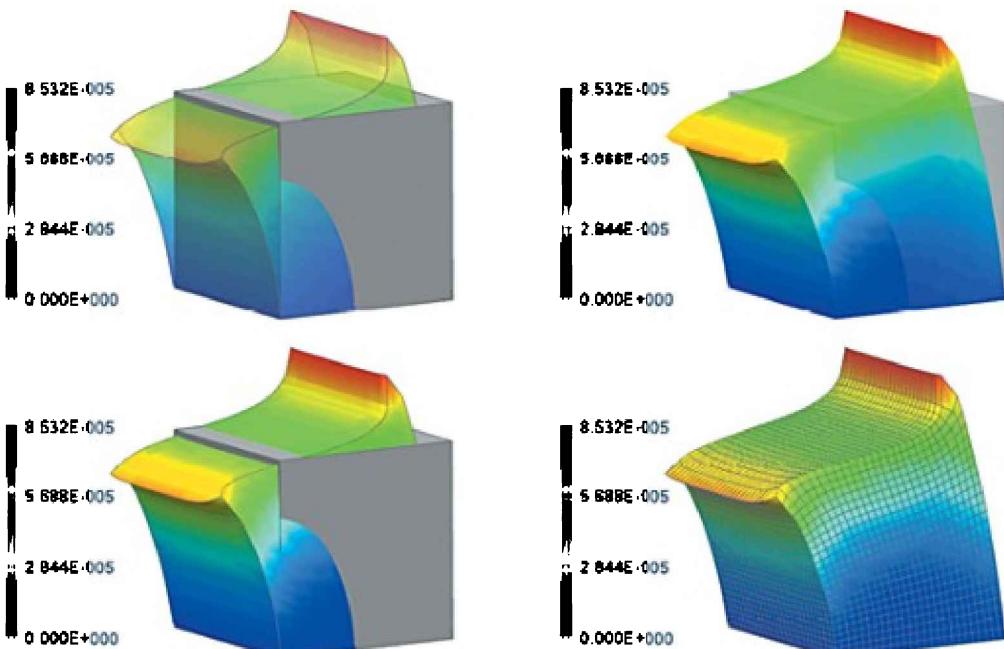


Figure 5.36. Displaying edges and faces differently in post processing

5.2.4. Displaying minimum and maximum values

There are two ways to display nodes with the minimum and maximum

values in the graphics area:

- Use the **Marker On/Off** command on the **Post Processing** toolbar.
- Select the **Show min-max markers** option in the **Annotation** tab of the **Post View** dialog box.
- To reposition the minimum and maximum result markers in the graphics area, use the **Marker Drag** command.

You can set up the markers display using the following options (Figure 5.37):

- Select the **Marker Fill Color** to set up the background color of the marker.
- Select the **Marker Line Color** to set up the color of symbols.
- Select the **Text Color** to set up the color of the header and palette values.
- Select the **Automatic Font Scaling** check box to modify the font size of markers, the header and the palette using a slider.

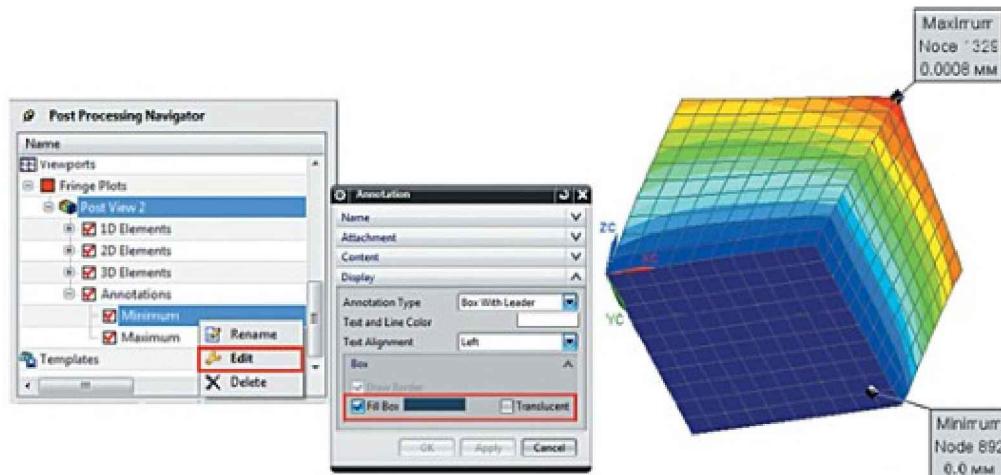


Figure 5.37. Displaying minimum and maximum values

5.3. Additional display features

This section describes tools you can use to display and analyze results for 1D beam elements and 2D shell elements, as well as options of the command you can use to animate simulation results.

5.3.1. Beam element section postprocessor

Standard results display tools do not allow displaying value distributions in sections of beam elements. For this purpose you need to use the separate **Cross-Section view** command on the **Post Processing** toolbar.

To display stress and strain distributions in cross sections of beam elements, you need to write loading conditions for these elements into the results file. To do this, in the *Output Requests* group, specify a *Force* value before you run the simulation.

Displaying a distribution of values in beam element cross section involves the following steps (Figure 5.38):

1. Use the **Cross-Section view** command.
2. Specify the beam element and the node in one of the following ways:
 - *Selection Mode – From Model*: select a beam element and one of its two nodes in the graphics area.
 - *Selection Mode – By Element ID*: in the dialog box, specify the number of the element (*Element*) and one of its two nodes (*At End – Aft or Fore*).
3. If necessary, you can preview the results by selecting the *Preview* check box. A small graphics window opens, which shows the von Mises equivalent stress distribution in the section.
4. Click **OK**.

In the graphics area you can see the von Mises stress distribution for the selected cross section, while in the results tree of the **Post Processing Navigator** toolbar, a new **Cross-Section View** node appears.

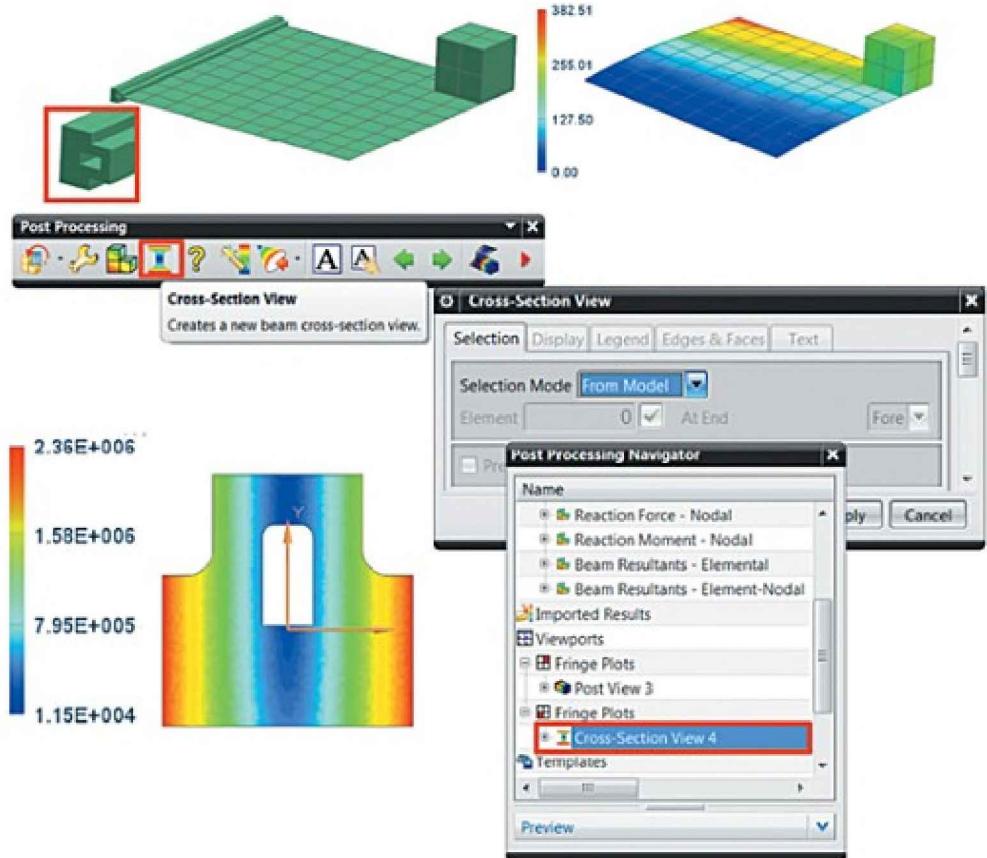


Figure 5.38. Displaying results in beam cross section

To analyze cross-section results, you can use the following commands that are slightly different from the standard *Post View <n>*:

- The ***Edit Post View*** dialog box contains an extra *Selection* tab where you can change the element and the node. You can use the *Display* tab to modify *Color Display* and use the ***Result...*** command. The other tabs do not differ from the description in para. 5.2.
- The ***Identify Results*** command allows to output numerical data in the cross section.
- The ***Marker On/Off*** command and the ***Marker Drag*** command display the maximum and minimum values in the cross section.

Note that value distributions in beam cross sections are displayed in the beam coordinate system, so you cannot change the coordinate system for

display.

5.3.2. Displaying results for shell elements

You can use an extra result presentation command for shell elements on the **Post Processing** toolbar, namely, the **Backface Culling** command. It displays results only on the “top” side of the shell elements, that is, on the face of the element with positive direction of the normal vector (Figure 5.39).

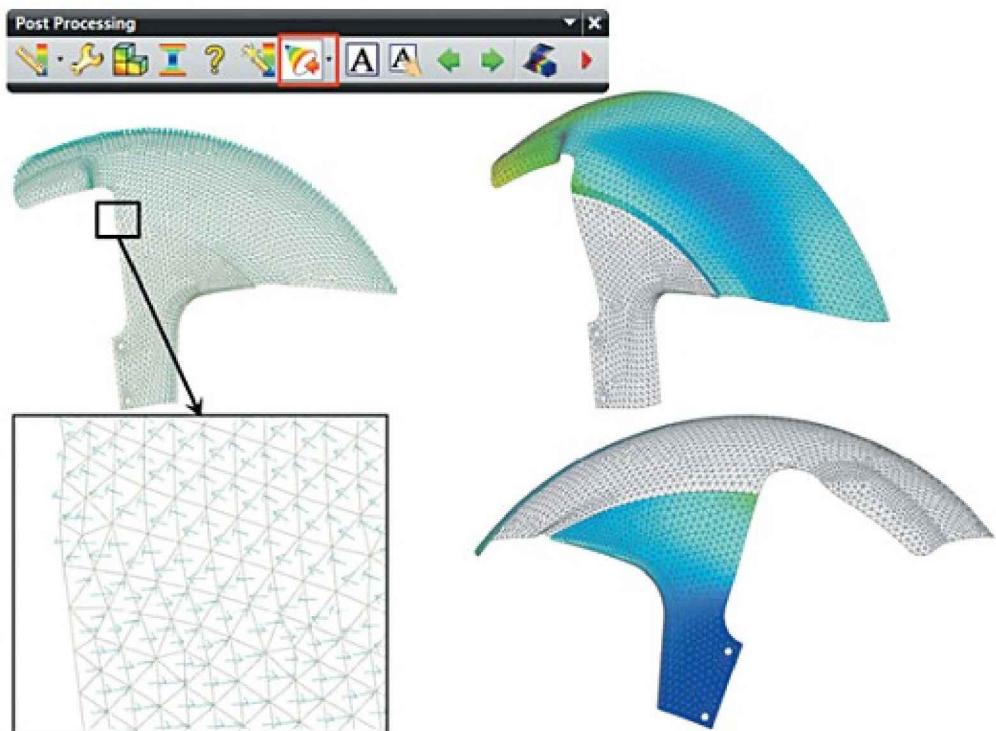


Figure 5.39. Displaying results for a shell using the **Backface Culling** command

This command is useful when you need to select the results display type for shell elements on internal (*Bottom*) and external (*Top*) sides of shells at the same time. To set up this display type, in the **Set Result** dialog box of the **Post Processing** toolbar, under **Shell**, select **Top and Bottom**. Note that simultaneously displaying results on both sides of shells is not meaningful when the **Backface Culling** command is not selected (Figure 5.40, C).

Here is an example of results display for shell elements. Figure 5.40 shows the following display types:

- A. Display on the external side of the shell only with the *Shell – Top* option.
- B. Display on the external side of the shell only with the ***Backface Culling*** command selected.
- C. Display on external and internal sides of the shell with the *Shell – Top and Bottom* option, ***Backface Culling*** command not selected.
- D. Display on external and internal sides of the shell with the *Shell – Top and Bottom* option, ***Backface Culling*** command selected.

The C variant is incorrect.

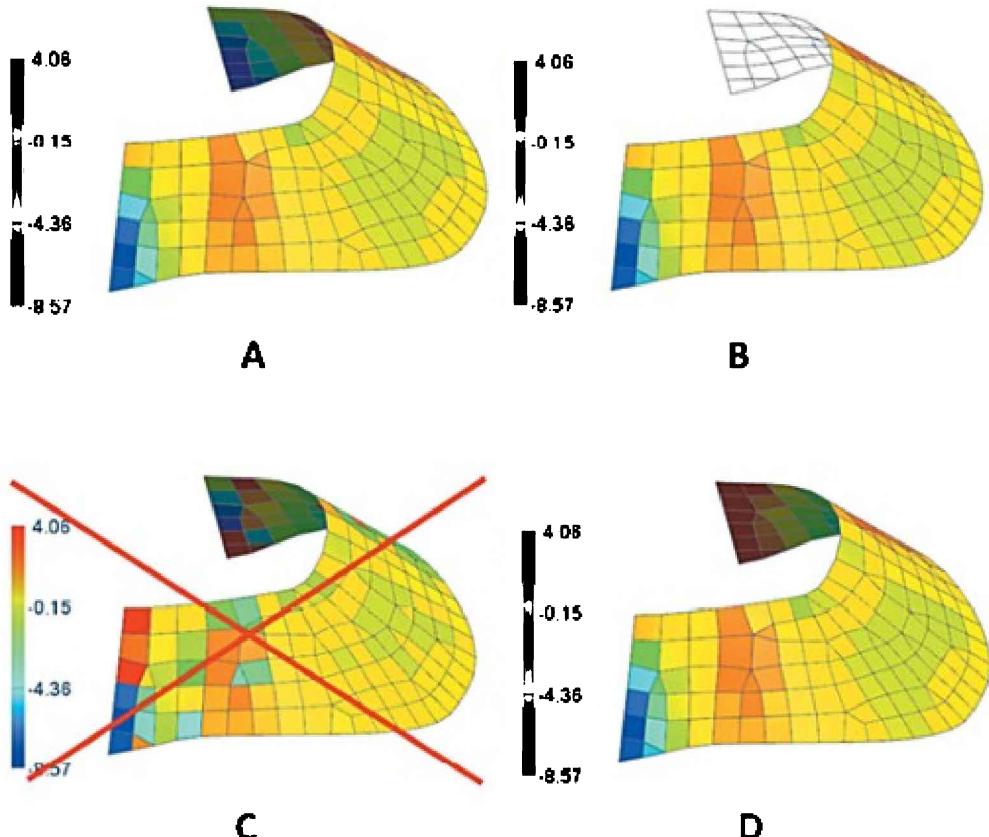


Figure 5.40. Displaying a single component of the stress tensor for the shell

5.3.3. Creating animations

Many problems are solved in a sequence of simulations, such as time steps for unsteady problems, iterations for nonlinear static problems, and so on. These are called *increments* in the results tree. To visualize the evolution of the model state (deformed, thermal, and so on) increment by increment, you can use the ***Animation*** command on the Post Processing toolbar. This

command creates an animation for one or more increments (Figure 5.41). There are two types of animated states that you can manage directly in the dialog box using the *Animate* option:

- Select *Result* to animate the result of a single increment or step as the value changes from 0 through the current value at the given step. *Style – Linear* corresponds to linear change (typically used for static problems), while *Style – Modal* corresponds to harmonic-law change and is used for natural frequency and mode calculation problems. You can type the number of frames in the *Number of Frames* box (Figure 5.41, A).
- Select *Iterations* to animate the selected result type in accordance with the values at each incremental step (Figure 5.41, B). To create the animation, specify the initial step/subcase and the initial increment, the final step/subcase, and the final increment. If there is a large number of increments, for example, in dynamic analysis, you should specify a *Step* value to skip intermediate increments.

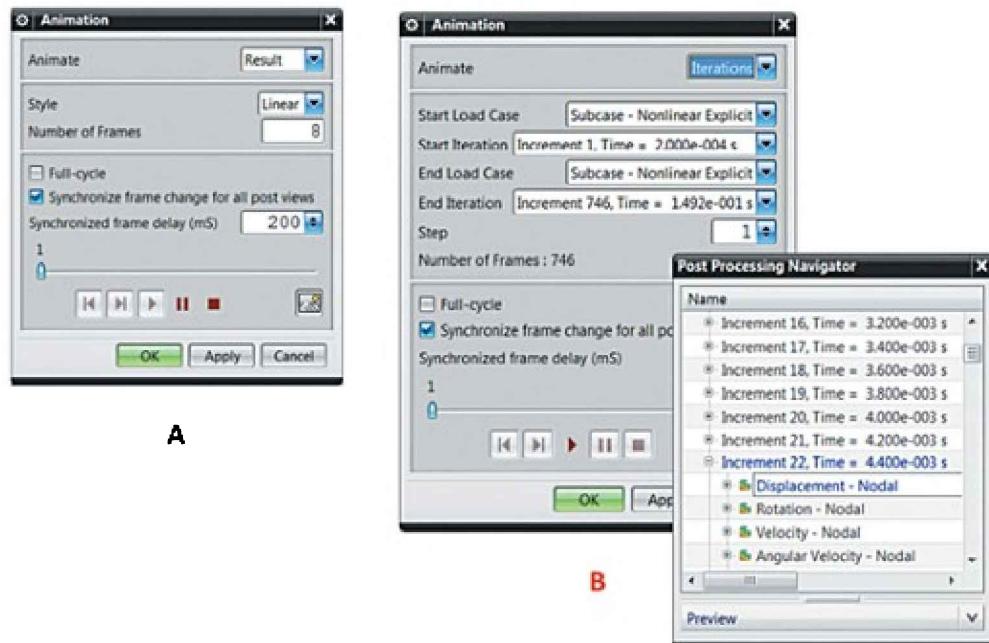


Figure 5.41. Creating animated states

You can specify the following additional parameters:

- Select the *Full Cycle* check box to create a cyclical animation,

running from the first increment to the last and back to first again.

- Enter the *Synchronized frame delay* to control the duration of each frame of the animation.
- The slider shows the current frame number.
- Use the control buttons to pause, stop, play, and switch frames.

To save the animated state, use the **Capture animated GIF** command or click **OK** after choosing →→**File**→**Export**→**Animated GIF** in the main menu.

When the command is completed, you can watch the animation or use **Post Processing** toolbar commands to stop, pause, play, and switch frames.

To view the results of different increments instead of creating animations, you can quickly switch between increments or modes using the **Previously Mode/Iteration**, or **Next Mode/Iteration** commands on the **Post Processing** toolbar.

5.4. Numerical output

In result analysis and processing, directly working with numerical values of different variables is particularly important. You can use the **Identify Results** command of the **Post Processing** toolbar to obtain results data as numerical values (Figure 5.42). You can use this command to:

- Display the value for a selected node set and corresponding elements directly in the graphics area.
- Create files of different formats containing values with numbers and coordinates of corresponding nodes.

When you use the **Identify Results** command, the **Identify** dialog box opens. The numerical values you obtain correspond to the particular variable that is currently selected for display. There are three logical parts in the dialog box: object selection parameters (Figure 5.42, A), the text area with data for selected objects (minimum and maximum values, sum of values, mean value, and numbers of nodes/elements) (Figure 5.42, B), and file output options (Figure 5.42, C).

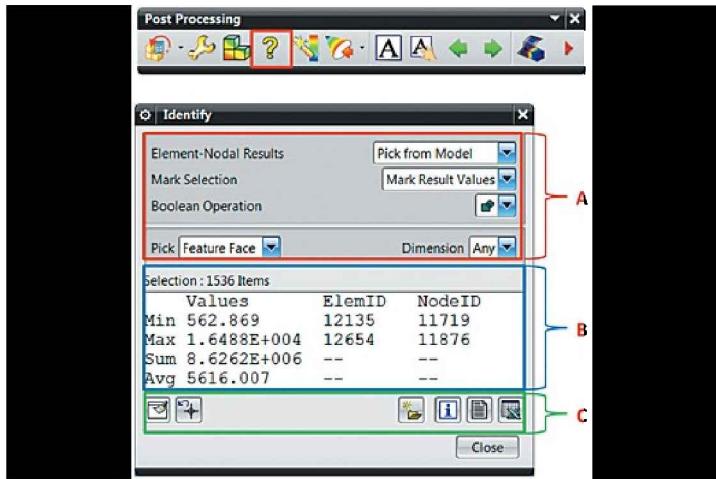


Figure 5.42. General view of the *Identify* dialog box

When you select nodes or elements, they are marked on the model in the graphics area. Use the *Mark Selection* option to control the display of the marks:

- Select *Mark Result Values* to display the value at each node.
- Select *Mark IDs* to display the number of each node.
- Select *No Marks* to turn off the data display.

Use the *Nodal Results* option to choose the node selection method:

- Select *Pick from Model* to select nodes directly on the model in the graphics area. With this method, you can select additional parameters:

Under *Pick*, specify the node selection method.

Under *Dimension*, set up the finite element size filter.

- Select *By Node IDs* to select nodes or elements by entering their numbers.
- Select *By Result Range* to select objects with values of the variable that fall inside the specified value range (*Below*, *Above*).
- Select *N Max Result Values* and *N Min Result Values* to automatically select *N* nodes with maximum and minimum values respectively. You need to specify the number of nodes <*N*>.
- Select *Nodes in Group* to select nodes that belong to the specified group.

You can use the ***Clear Highlights*** and ***Clear Selection*** commands in the bottom left part of the ***Identify*** box to quickly clear the information display in the graphics area and reset the object selection correspondingly.

You can use the ***Boolean Operation*** option to perform logical operations on selected node sets.

You can use the following commands to output the information for selected nodes and elements located in the bottom right part of the dialog box:

- Use the ***List Selection in Information Window*** command to output the information into a new window where it can be saved as a text file.
- Use the ***Export Selection to a File*** command to output the information to a text file with a user-specified delimiter. When you use this command, a new ***Identify File Export*** dialog box opens. In this dialog box, you can specify the types of data to export, the ***Delimiter*** to use for the decimal part and the number of digits in the decimal part (Figure 5.43, A).
- Use the ***List Selection in Spreadsheet*** command to export all data into an Excel spreadsheet, which is then opened. Note that in addition to numerical values and numbers of nodes or elements, the information in the graphics area header is also written to the spreadsheet (Figure 5.43, B).

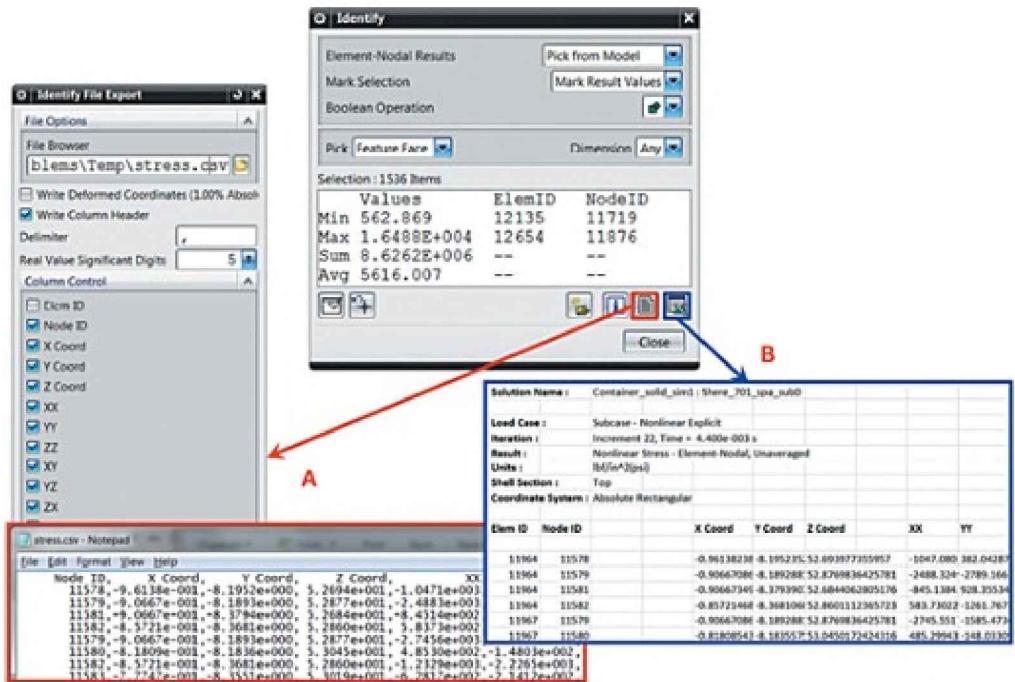


Figure 5.43. Exporting numerical data into the table

5.5. Working with graphs

In most cases when you analyse results, displaying the distributions of values in the model as contour plots is not enough. If you need to process numerical values, you can use the **Identify** command described in para. 5.4, create a data file, and post process it, for example, draw charts in Excel. This method is, however, time-consuming so you would typically use NX features to create, edit, and analyse graphs. The **New Graph...** command, which you can access by right-clicking the created *Post View <n>*, can be used to:

- Create graphs representing the variation of the displayed value with node number or location (along the selected path).
- Create graphs representing the variation of values at specific nodes depending on the increment number (time).

The graphs you create can be overlaid, edited, or exported. You can perform mathematical operations on them and change their display type. NX **Advanced Simulation** has a rich set of features for working with graphs. Only the most important tools are described in this book.

5.5.1. Creating graphs

To create a graph along a path, specify the path using the **New Path...** command, which you can call by right-clicking the corresponding *Post View <n>*. You can use this command (Figure 5.44) to create a new path through node numbers (*Node ID*) or coordinates (*Coordinates*). You need to specify the number of intermediate points (*Insert N points per segment*). Specify the initial and final points of the path for the selected *Method*. After this, the *Paths* node appears in the results tree with the list of created *Paths*.

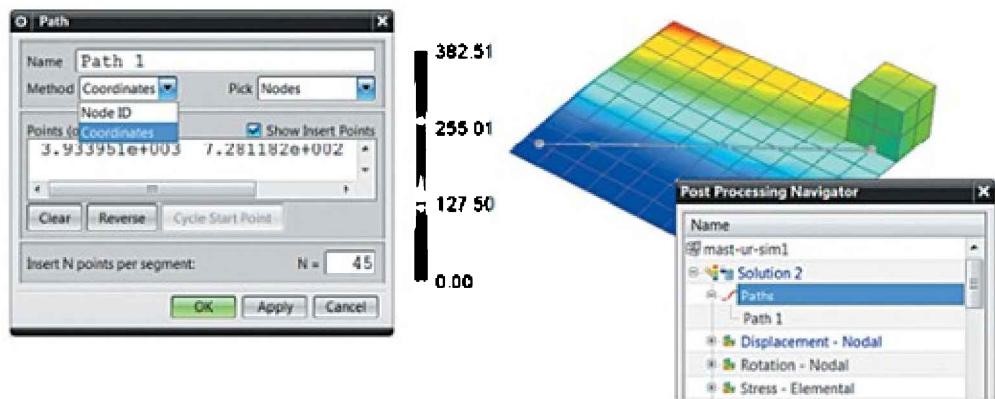


Figure 5.44. Creating a path for building a graph

The layout of the **New Graph...** dialog box depends on the selected *Graph Type*. Here is a description of the key parameters for creating two types of graphs:

1. *Extract On Path* creates value variation graphs along a specified path (trajectory) (Figure 5.45, A). In this case you specify the *Path*, select the type of values to plot on the *Abscissa*. If a node contains more than one value (for example, if the node belongs to elements with different materials), specify the value to use for the *Value Sharing* option: mean, minimum, or maximum.
2. *Across Iterations* creates value variation graphs depending on increments (time) (Figure 5.45, B). Specify the node number or coordinate in the *Track list*, then select the node by directly entering its number or coordinates in the *Node ID* box or by clicking the node on the model. Select the initial and final increment and the necessary *Step*. Select the type of values on the *Abscissa*: point in time or increment number. If necessary, maintain the *Value Sharing* parameter.

If the *Plot graph on creation* check box is set, the command displays a graph or creates a *Graphs* node containing all your graphs.

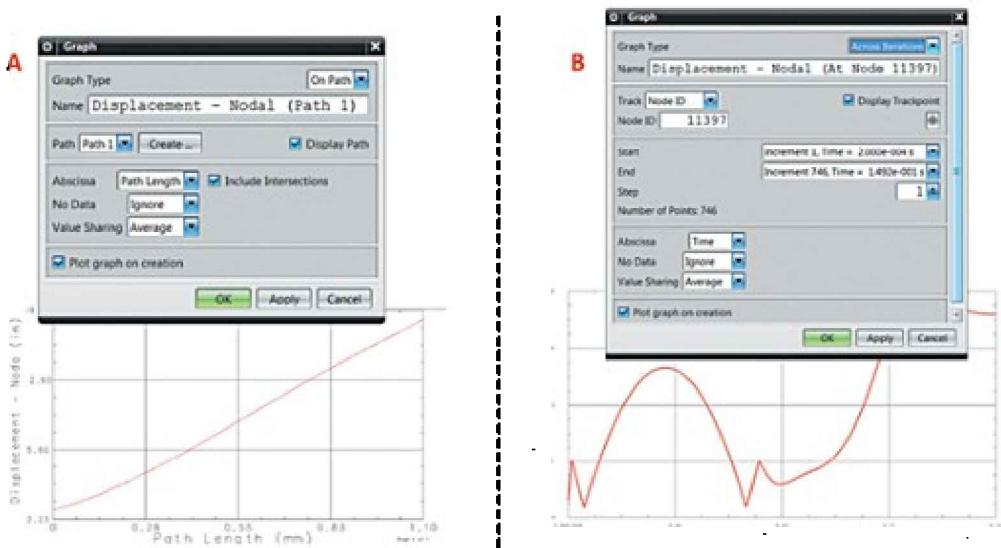


Figure 5.45. Creating graphs of values that vary with path and time

5.5.2. Managing graphs

The shortcut menu of each graph contains the following commands:

- Use the **Edit** command to edit numerical values of the graph using the **XY Function Editor** command. Generally, this command is used to create new graphs. It has a rich graph editing feature set.
- Use the **Copy** command to copy (or add) the graph to an AFU (Associated Function Utility) file. This file is associated with the simulation model. It contains all tabular data used in the model. These data are managed, plotted, and processed in the **XY Function Navigator** tab of the Resource Bar.
- Use the **Delete** command to delete the graph.
- Use the **Plot** command to display the graph in the graphics area.
- Use the **Overlay** command to plot the graph over another graph displayed in the graphics area (Figure 5.46).
- Use the **List** and **Information** commands to output all values and other information in text form.

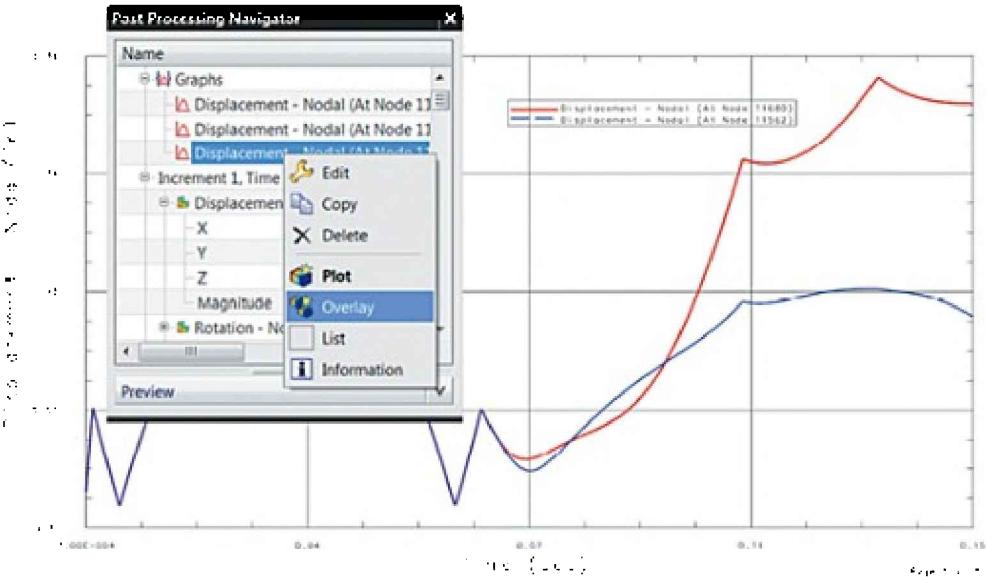


Figure 5.46. Overlaying graphs

5.5.3. Working with graphs. XY Graph toolbar

In addition to the previously described graph commands, you can use other features on the **XY Graph** toolbar. These include reading numerical values, complex transformations, and graph layout options (Figure 5.47).

The **XY Graph** toolbar commands belong to four logical groups: graph layout, value reading, graph area scaling, and switching between existing graphs. A brief description of these commands follows:

1. The **Edit** drop-down list contains commands (Figure 5.47, A) that you can use to edit: display format of axes, labels, the graph grid, curves, and so on. You can also use these commands to manage the display of complex values (real/imaginary components or amplitude/phase). You can save all graph display settings in a template that can be reused for other graphs.
2. You can use commands in the **Probing Mode** list to display values at points by directly specifying the points on the curve using one of the available methods (Figure 5.47, B).
3. To scale the graphs, that is, to edit the value output range, you can use commands in the **Zoom by X Only** list (Figure 5.47, C).
4. If you have several graphs, you can switch between them using the **Previous Page** and **Next Page** commands.

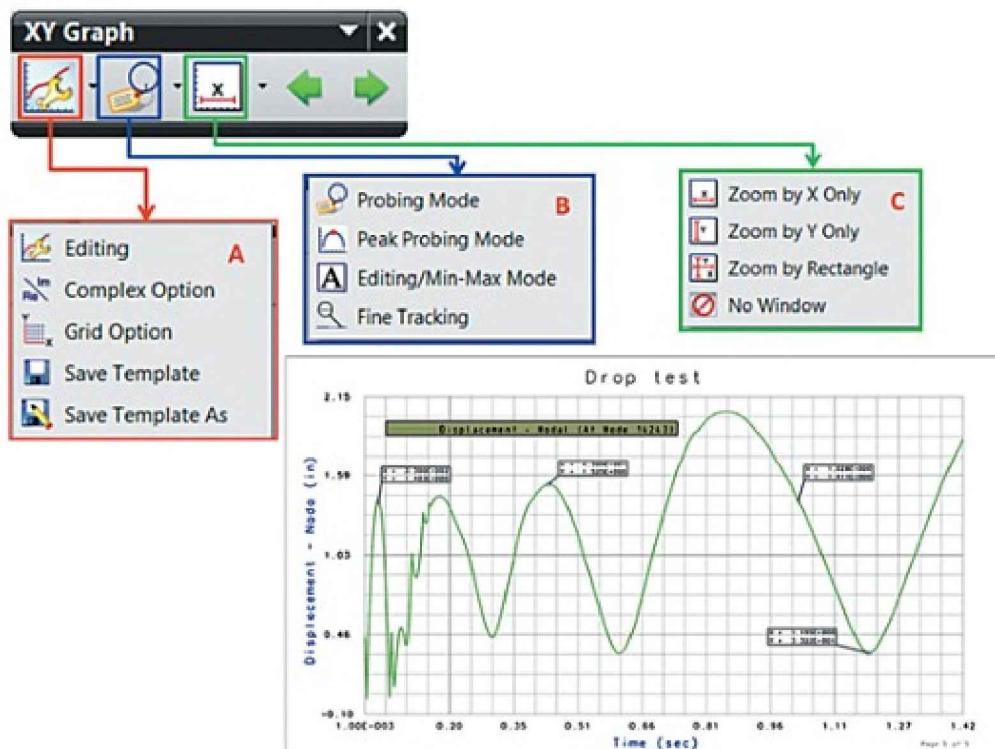


Figure 5.47. Commands of the XY Graph toolbar. Sample graph layout

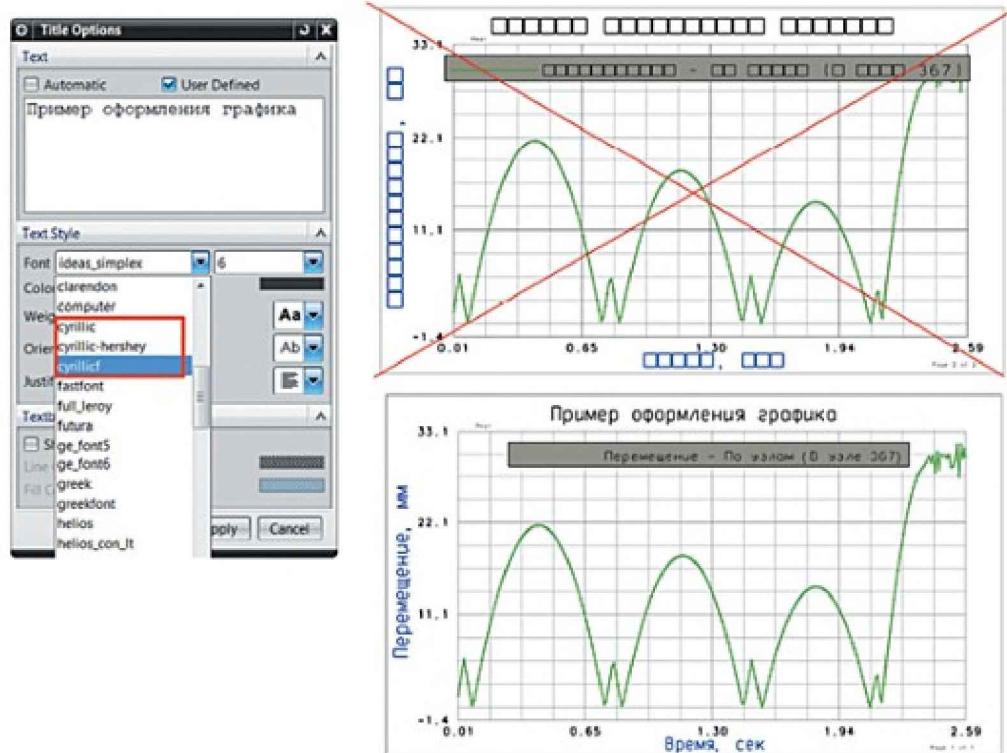


Figure 5.48. Changing graph label font to display special characters

5.6. Additional result analysis tools

5.6.1. Free body results

When you analyse results for some problems, you need to output the so-called free body results to display total internal forces and moments in a part of the model, totaled up relative to a specified point. You can do this using the **Free Body Results** command. Right-click the corresponding *Post View <n>* and choose the command in the shortcut menu. Remember that in this case you use *Grid Point Force* nodal forces that must be included in the result set before you run the solution.

5.6.2. Manipulating results

In some cases you need to work with several result sets for different sub steps and solutions simultaneously. You can use the **Results Manipulation** command of the **Post Processing** toolbar to create new results based on original results (Figure 5.49).

Remember that you need to manually verify some parameters of the processed result sets. The quantity and numbering of nodes and elements must match, the models that are used to obtain the results must be identical, and so on.

The **Results Manipulation** command yields a *Universal format file (*.unv)*. You can set up the output of this file in the *Output* group by setting the following options:

- In the *Action* list, select one of the output methods: export results, export results with FE model, create with simultaneous import, add to an existing results file with an FE model or without one.
- Use the *Incompatible Results*, *No Data*, and *Evaluation Error* options if results are incompatible, or some data are missing when you create the new results output file.
- In the *Name* box, type the name to use when the new results file is subsequently displayed in the **Post Processing Navigator** tab.
- In the *Universal File* box, specify the location and name of the new results file.

Here is a description of two available manipulation types: *Envelope* and *Combination*.

Select the *Envelope* type to compare several results for a specified value component, and output the minimum or maximum scalar value at each node/element (Figure 5.49, A). The values can correspond to different sub steps, increments, and solutions. They must, however, belong to the same FE model. To use the command with the *Envelope* option, do the following:

- Specify the *Type*.
- In the *Result* list, select one of the results loaded in NX.
- Specify a value for the *Result Type* list.
- Select a component of that value in the *Component* list.
- Select a *Coordinate System* and *Units*.
- Add the value to the list by clicking the corresponding button.
- If necessary, add more values corresponding to other results to the list of the same type.
- In the *Operation* group, select the minimum and maximum values for list items.
- Specify the output options and the name of the new results file.

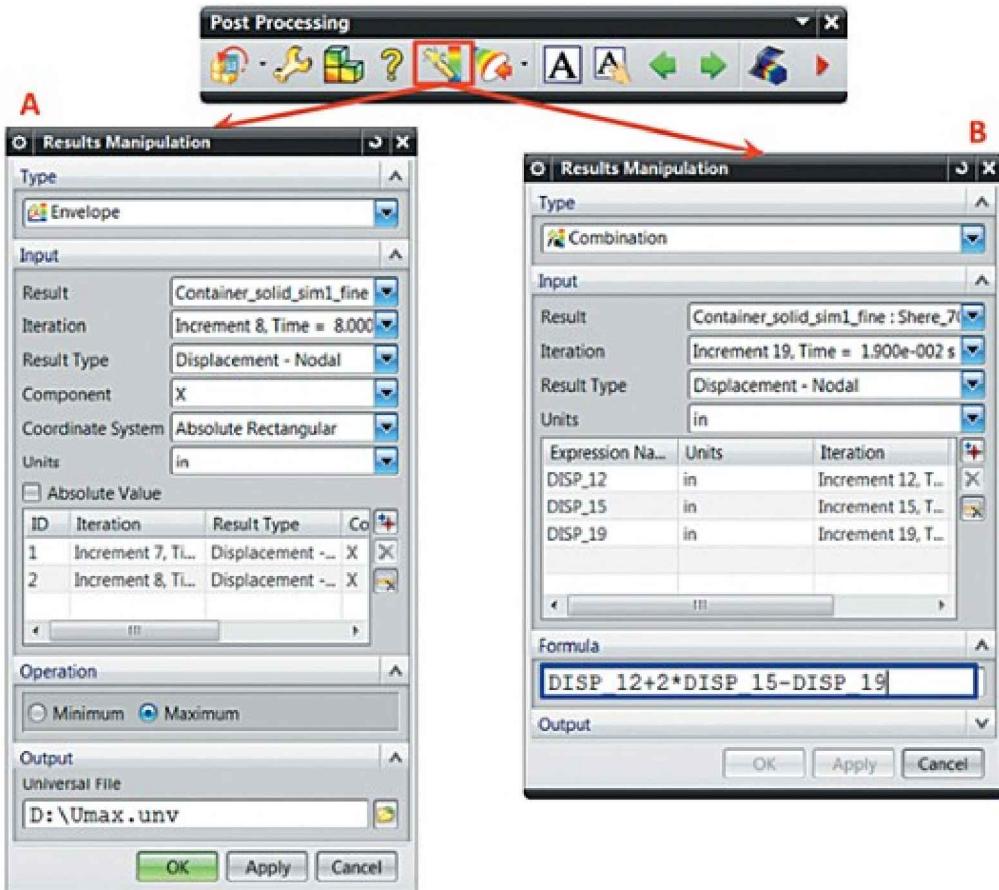


Figure 5.49. Manipulating results

You can use the *Combination* type to perform mathematical operations (using NX syntax) on several results with the same solution type. A parameter is automatically assigned to the specified results type. You can use this parameter when you compose the formula (Figure 5.49, B). The values you select may correspond to different sub steps, iterations, and solutions, but they must have the same solution type, the same FE objects and the same value type. If you use vector or tensor values, you can specify only the value itself, the result of the command extends to all components. To use the command with the *Envelope* option, do the following:

- Specify the *Type*.
- In the *Result* list, select one of the results loaded in NX.
- Specify a value for the *Result Type* list.
- Add the value to the list by clicking the corresponding button.
- If necessary, add more values corresponding to other results to

the list of the same type.

- In the *Formula* box, enter the mathematical formula based on parameters assigned to each value by the software, for example $p36+p37$;
- Specify the output options and the name of the new results file.

5.6.3. Exporting images

When you prepare reports or presentation materials with simulation results, you need to use the content of the graphics area. To save it, choose the main menu command →→**File**→**Export**→**PNG/JPEG/GIF/TIFF/BMP**, which saves the content displayed in the graphics area as an image of the selected format.

To save the distribution of values of the selected variable, you can export it into a *JT* format file. You can do this by right-clicking *Post View* and choosing **New JT File...** in the context menu.

Chapter 6. Modelling composite structures

NX Laminate composites (NX LC) is a module for NX Advanced Simulation that allows you to quickly design and analyse laminate composites in various ways. The composite model created in NX LC and the corresponding finite-element simulation model reflect all geometry and physical aspects of the composite product.

During initial design in NX LC, you can optimize the composite structure locally and globally for the given loading conditions.

NX LC also supports data exchange with Siemens' Fibersim, the leading software product for modeling composite products. You can use this integration to improve your model created in NX LC using Fibersim functionality and perform a verification simulation using NX Nastran, MSC Nastran, ANSYS, ABAQUS, or LS-Dyna.

By using NX LC tools to view and analyze results, you can discover danger areas on each ply based on various criteria, and also comparatively analyze simulation results against experimental data.

6.1. Special aspects of working with composite structures in NX

Composite structure modelling and simulation involve a number of standard steps:

- Creating or importing geometry into NX
- Creating the FE file and the simulation file with the selected solver and solution type
- If necessary, simplifying or modifying the geometry
- Creating the FE mesh, taking into account physical properties.
- Creating the composite structure
- Specifying boundary constraints
- Solving the problem
- Viewing results and judging whether the structure is sound

Depending on the chosen solver and the method used to create the FE

composite model (para. 6.1.1), you can use element types listed in the following table:

Solver	2D elements	3D elements
NX Nastran and MSC Nastran	CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, CTRIA6	CHEXA, CPENTA
Abaqus	S3, S3R, S4, S4R, S4R5*, S8R, S8R5*, STRI3, STRI65	SC6R*, SC8R*
ANSYS	SHELL91, SHELL99, SHELL181	SOLID186 SOLID191*, SOLSH190*
LS-DYNA	ELEMENT_SHELL	ELEMENT_TSHELL*

* Available only for physical properties-based creation method.

This chapter describes creating composites and special aspects of processing them for NX Nastran. Other solvers are not covered.

6.1.1. Methods of creating composites and their arrangement in NX

Modelling composite structures implies building a 2D FE mesh on the surface of the CAD model or creating 3D elements. Surfaces used to build the FE model are analogous to jigs used in composite product manufacturing. The FE model you create is used to produce a preliminary draft of the composite product by determining properties, positions and orientation of plies. You can use NX Laminate composites to create composite products in one of two ways:

1. Specifying physical properties using the **Laminate Modeler** tool and assigning these properties to corresponding FE meshes. This method involves performing the following steps:
 - Create a 2D shell or 3D FE mesh
 - Assign material orientation
 - Create composite physical properties for the mesh
 - Validate the composite

If you use this method, all information about the composites is specified in physical property tables (Figure 6.1, A), which you can create, edit, copy, or delete. You can also edit material properties after creating the composites.

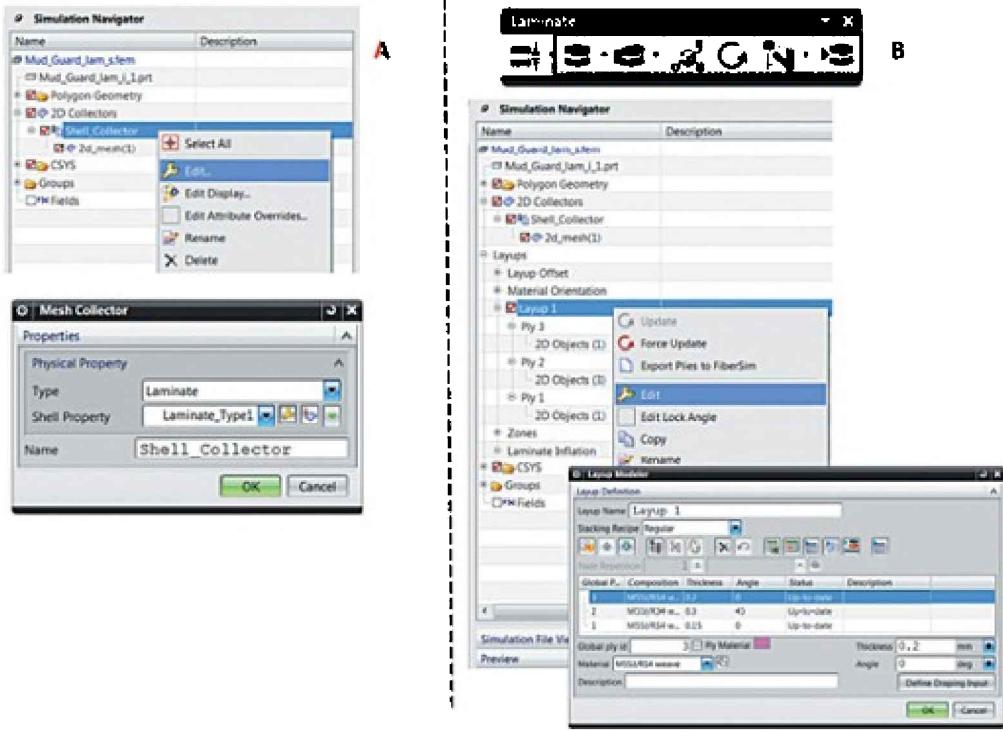


Figure 6.1. Working with composites

2. Draping using the ***Global Layup*** tool. In this case, you assign so-called global plies with a specific layup method to geometry faces. After that, you simulate zones with unique properties of composite materials. This process of composite creation involves the following steps:

- Create a 2D shell FE mesh
- Assign material orientation
- Create physical properties of the composites with reference to the global layup
- Specify the global layup and specify the material
- Select draping method
- Create zones

Composites created in this way correspond to objects (plies, offset, material orientation, zones) that you can find in the model tree in the **Simulation Navigator** toolbar. You can also create, copy, edit, and delete global ply layups (Figure 6.1, B).

6.1.2. Terminology and coordinate systems

Before discussing composite creation methods in detail, it is necessary to establish some basic terms and the coordinate system used with NX Laminate composites.

Here is a list of common NX terms:

Composite or *Laminate* – a material consisting of several thin layers of material that can have different properties.

Ply – a layer of the composite consisting of a simple or compound material which is, in a general case, orthotropic.

Core – a relatively thick ply dividing other plies that is used to improve bending performance of the composite.

Global Ply – the ply that is managed in the **Layup Modeler** dialog box.

Ply Layup – the order of plies in the stack and their orientation that can be modified in **Laminate Modeler** or **Layup Modeler**.

Ply Material – a material with effective stiffness and strength properties derived from the properties of matrix material and fibers, taking into account the microstructure of the ply.

You can use the following types of coordinate systems to model composites:

- The *Element coordinate system* is defined by the location and numbering of the element nodes (Figure 6.2, A).
- The *Material coordinate system* does not depend on the FE mesh. It is used to orient properties of anisotropic and orthotropic materials, for example, composite materials (Figure 6.2, B).
- The *Ply coordinate system* determines the orientation of ply fibers (Figure 6.3) (axis 1).

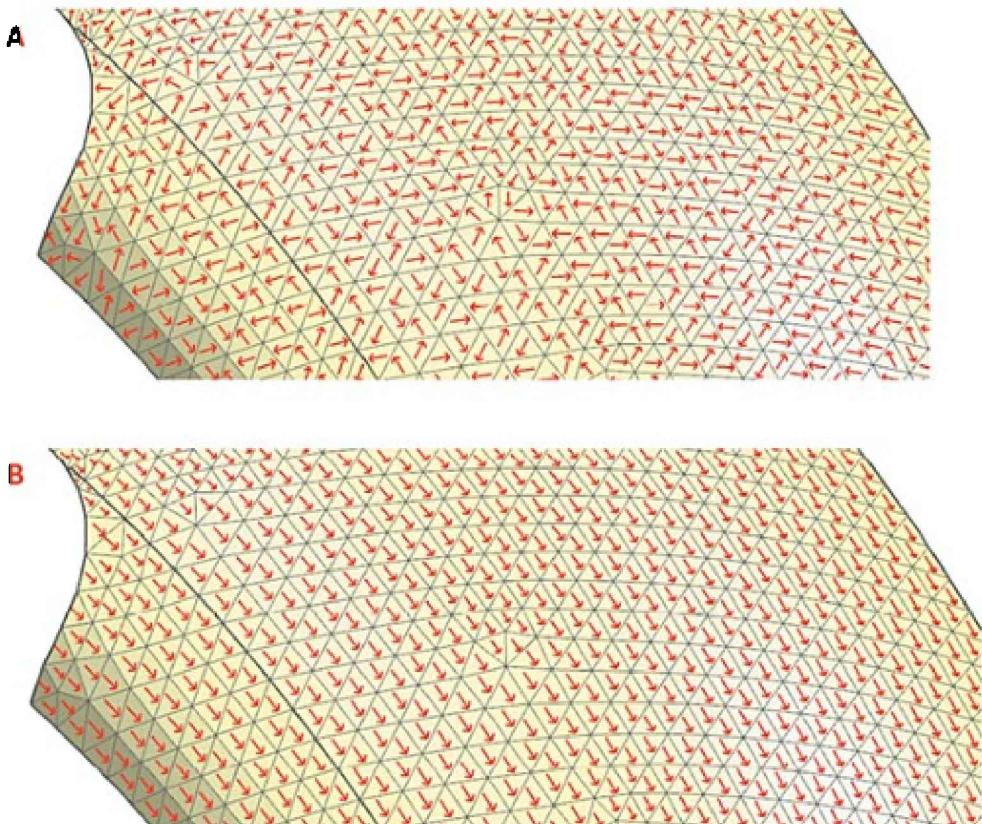


Figure 6.2. Coordinate systems

Figure 6.3 shows a composite ply layup. The XY plane location is defined by the reference plane. X and Y axes match the material orientation coordinate system that is called the *Laminate coordinate system*. The Z axis is directed along the normal vector of the composite. Axes 1 and 2 correspond to the ply coordinate system, axis 1 runs along the fiber in unidirectional-type plies, while in woven-type fabrics, it runs along the warp fibers. Axis 2 is directed along the normal to the fiber and corresponds to the weft direction in the woven ply. The θ angle is the angle between axes X and 1. The Z coordinate is responsible for the position of plies with regard to the reference plane.

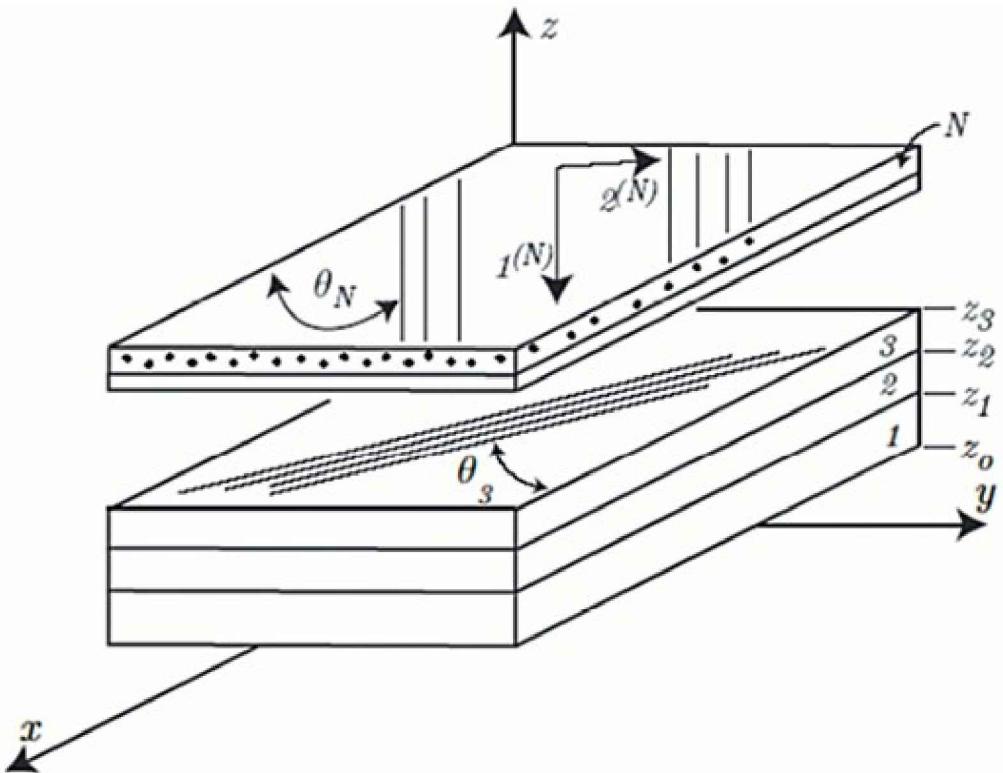


Figure 6.3. Composite and ply coordinate systems

6.2. Creating composites based on physical properties

Creating the composite by specifying composite physical properties for the FE mesh is the simplest method but its capabilities are limited. If you have an FE mesh, you need to set the material coordinate system and use the **Laminate Modeler** to create a ply layup and specify materials, thicknesses, layup angles, and other properties.

6.2.1. Material orientation

By default, the material coordinate system coincides with the element coordinate system so you need to reassign material orientation to define the composite coordinate system. Areas where material coordinate systems are oriented differently must correspond to different FE meshes.

You can set material orientation in the **Mesh Associated Data** dialog box, which you can access by right-clicking the FE mesh in the model tree (Figure 6.4, A). Select the orientation method in the *Material Orientation*

group: relative to the selected coordinate system (*MCID*); by specifying an angle between the new and the old OX axes (*Orient Angle*); by specifying the OX axis direction tangentially to a curve (*Tangent Curve*); by vector (*Vector*).

You can use the ***Finite Element Model Check*** command with the *Element Material Orientation* check to visualize and check the X axis direction of the material coordinate system (Figure 6.4, B).

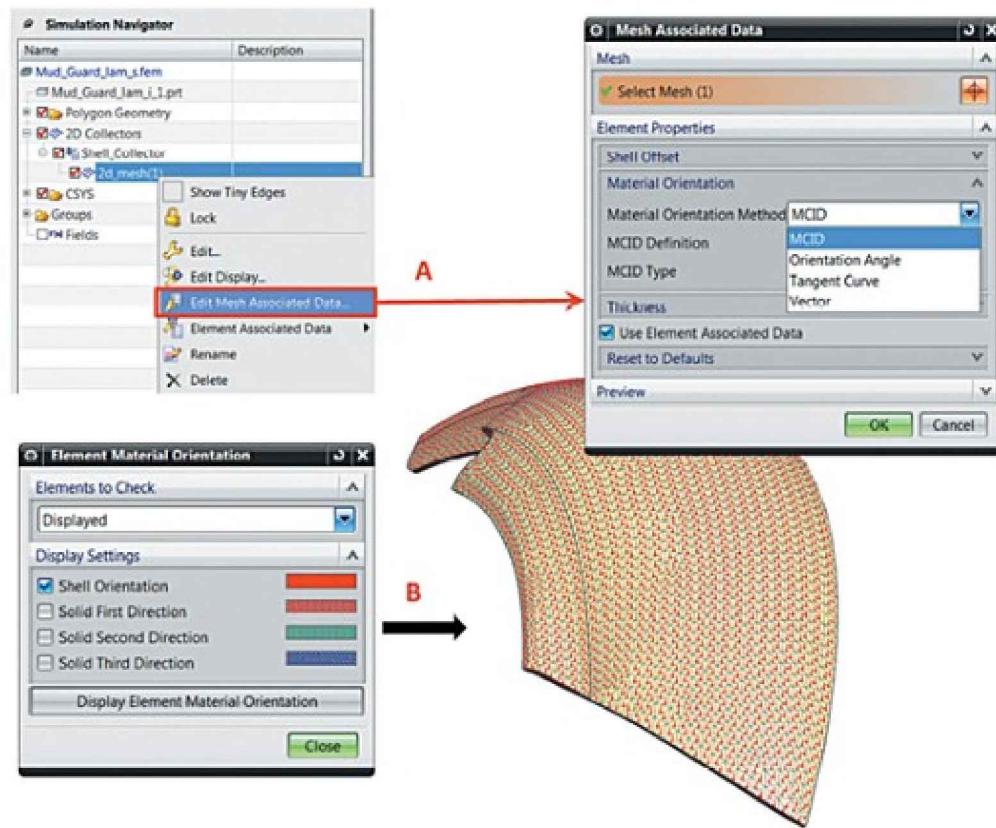


Figure 6.4. Setting up and displaying material orientation

6.2.2. Description of the Laminate Modeler

When you use this method, you assign properties of the composite material to a set of finite elements that share a collector. To do so, in the FE mesh collector shortcut menu, select ***Edit***. In the ***Mesh Collector*** dialog box, in the *Type* list, select *Laminate*. In the *Shell Property* box, select existing composite material properties from the list or specify new properties (***Create Physical...***). The ***Laminate Modeler*** dialog box appears.

You can also open this dialog box from the **Laminates** toolbar using the **Laminate Physical Properties** (Figure 6.5).

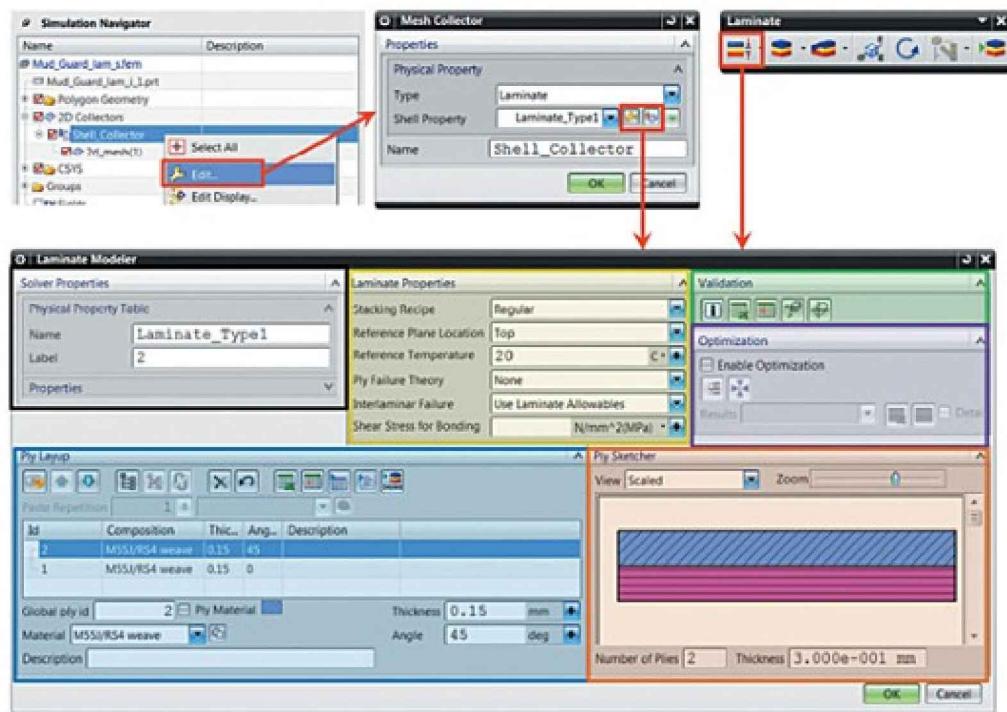


Figure 6.5. Using the *Laminate Modeler* dialog box and its general structure

The **Laminate Modeler** has six option groups (Figure 6.5):

- The *Solver Properties* group includes solution parameters and output options (see para. 6.5)
- The *Laminate Properties* group includes properties of the composite material and the layup method
- The *Ply Layup* group is used to create plies and ply groups with a specified layup method
- The *Ply Sketcher* group is used for visualizing the created layup
- The *Validation* group determines stiffness and strength parameters of the composite sample
- The *Optimization* group is used to find optimum properties of the composite

The following is a detailed description of options in each group:

The **Laminate Properties** group

The *Stacking Recipe* option governs the stacking sequence (Figure 6.6) of plies created in the *Ply Layup* group:

- The *Regular* option preserves the sequence specified in the *Ply Layup* group.
- The *Symmetric* option creates a symmetric stacking. You specify a ply sequence that is then mirrored.
- The *Symmetric with Core* option creates a layup symmetric with respect to the filler. In this case the first ply in the list in the *Ply Layup* group is the filler.
- The *Repeated* option repeats the layers created in the *Ply Layup* group.
- With the *Repeated with Core* option, the first ply is considered the filler, and all other plies are repeated with regard to that ply.
- The *Inherited from Layup* option references the global layup using the **Layup Modeler**.

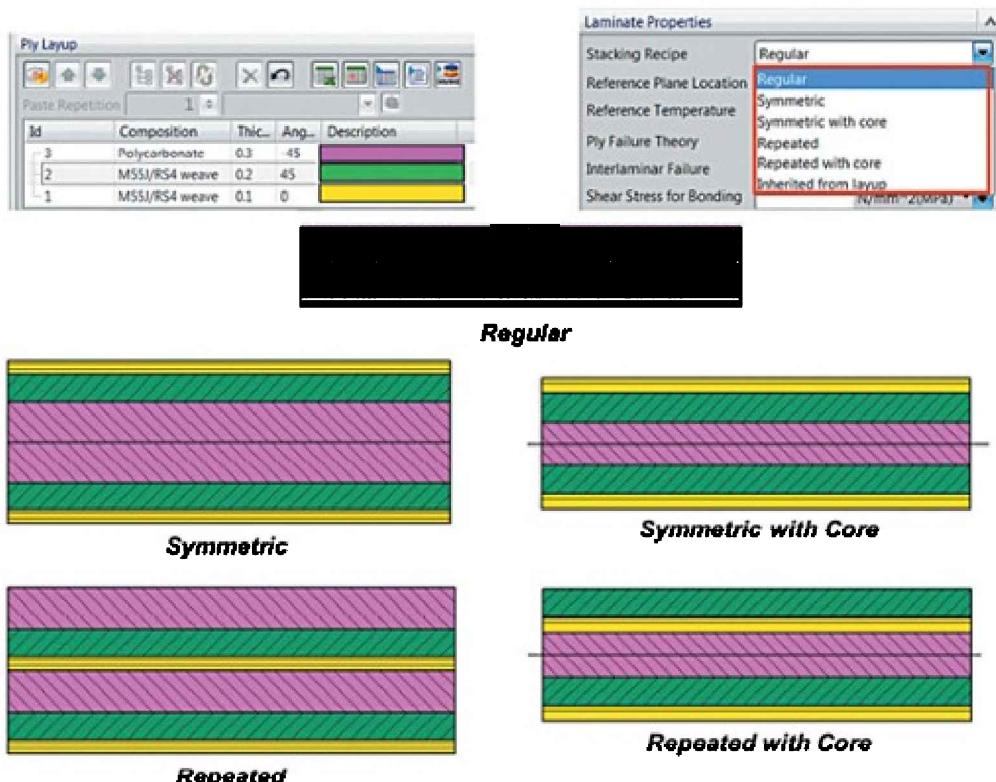


Figure 6.6. Ply layup types

The *Reference Plane Location* parameter defines the location of the ply

stack with regard to the XY plane of the composite coordinate system (Figure 6.7):

- Select *Top* to have the reference plane coincide with the top plane of the composite.
- Select *Middle* to have the reference plane coincide with the midplane of the composite.
- Select *Bottom* to have the reference plane coincide with the bottom plane of the composite.
- Select *Specify* to offset the reference plane from the bottom plane of the composite by a specified *Bottom Fiber Distance*. A negative value means the composite is offset with regard to the reference plane in the direction opposite to the direction of the normal vector.

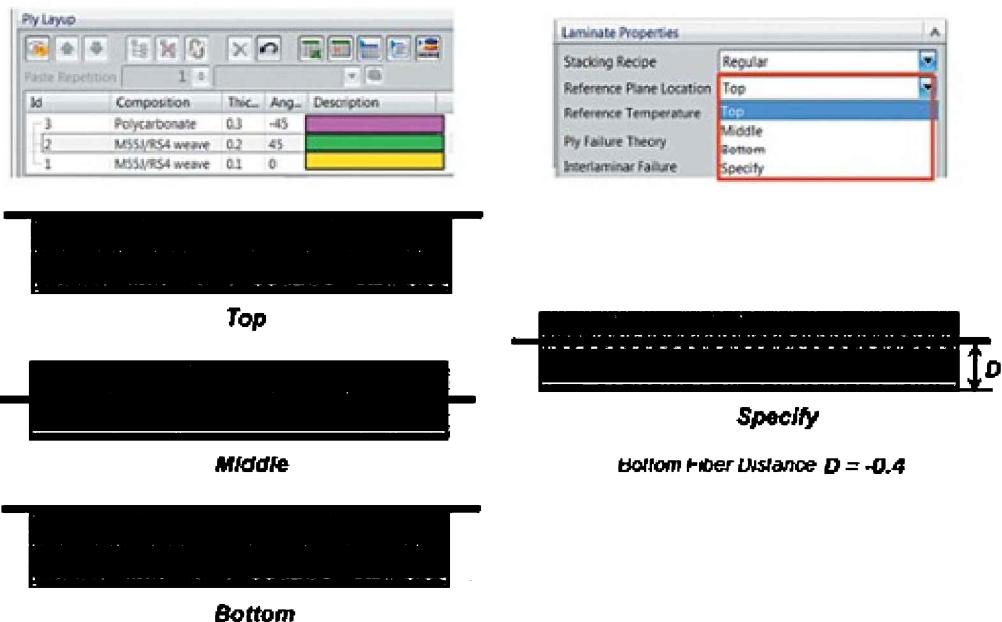


Figure 6.7. Positioning the composite with regard to the reference plane

The *Reference Temperature* parameter sets the temperature that is used to determine temperature-dependent material properties.

The *Failure Theory* option is used to select the failure criterion to evaluate strength properties of the composite:

- *Hill*
- *Hoffman*
- *Tsai-Wu*

- *Maximum Stress*
- *Maximum Strain*
- *Puck*
- *LaRC02*

The *Allowable Stress For Bonding* option defines the allowable stress value to evaluate delamination.

The *Ply Layup* group

This group contains ply creation and management tools. The plies you create and their parameters (thickness, material, and orientation angle) are displayed in table form (Figure 6.8).

To create a new ply, use the **Create New Ply** command and specify the following options:

- Select a *Material* from the list of available materials or from NX material library. Alternatively, you can use a ply material with microstructure (para. 6.4).
- Enter the *Thickness* of the ply.
- Enter the fiber orientation *Angle*.

Note that a *Global ply id* is assigned to the ply. You can change this number. Ply 1 is always the lowermost (*Bottom*) ply of the composite, and the ply with the largest number is the uppermost (*Top*) ply of the composite.

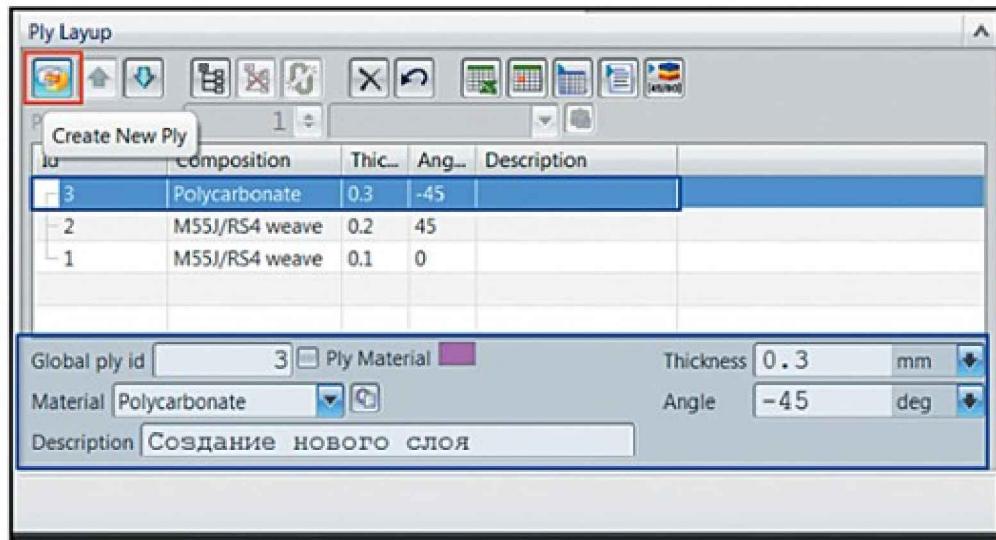


Figure 6.8. Creating plies

If you need to create a repeating structure, use the **Group Selected Plies** command. You can use the shortcut menu to copy and paste a group of plies before or after a selected ply in a number of ways: *Paste Above*, *Paste Below*, *Paste Symmetric Above*, *Paste Symmetric Below*, *Paste Anti-symmetric Above*, *Paste Anti-symmetric Below*. Ply sets created in this way are coupled with the initial set, that is, any ply parameters edited or added in the initial set will be replicated in the copied sets. If you need to decouple the sets, use the **Unlink Selected Ply Group**.

Figure 6.9 shows the steps you need to take to create a ply group and make two copies of the group.

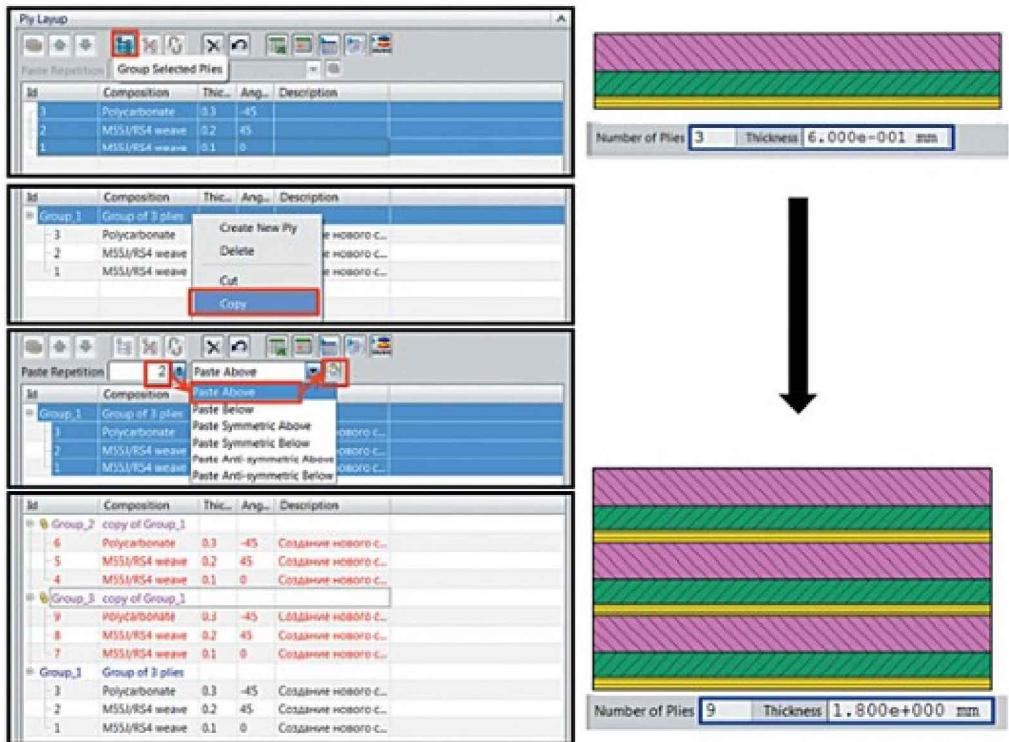


Figure 6.9. Working with ply groups

So, to create and edit ply layups, you need the following tools:

- The **Create New Ply** command.
- Commands for copying sets (groups) of plies and pasting them with regard to a specified ply.
- The **Move Selected Ply Up/Down** command.
- The **Delete Ply** command.

In addition, the *Ply Layup* group provides auxiliary commands for fast creation, export, and import of ply layups:

- You can use the **Import Layup Using Shorthand Format** command to quickly create plies by entering data in a particular format as a *Shorthand String* (for example, [(0/45/90)_s]) with the same thickness and material. These parameters can be changed for each ply in the ply table.
- You can use the **Export/Import Layup to/from Spreadsheet** commands to efficiently process many plies by editing an *Excel* spreadsheet. You can also use similar commands to export and

import CSV files (comma-separated values).

The *Ply Sketcher* group

This group is informational. It displays a layup scheme with accurate representation of dimensions and orientation angles of ply fibers (Figures 6.10). It also shows information about the total *Thickness* of the composite and the *Number of Plies*.

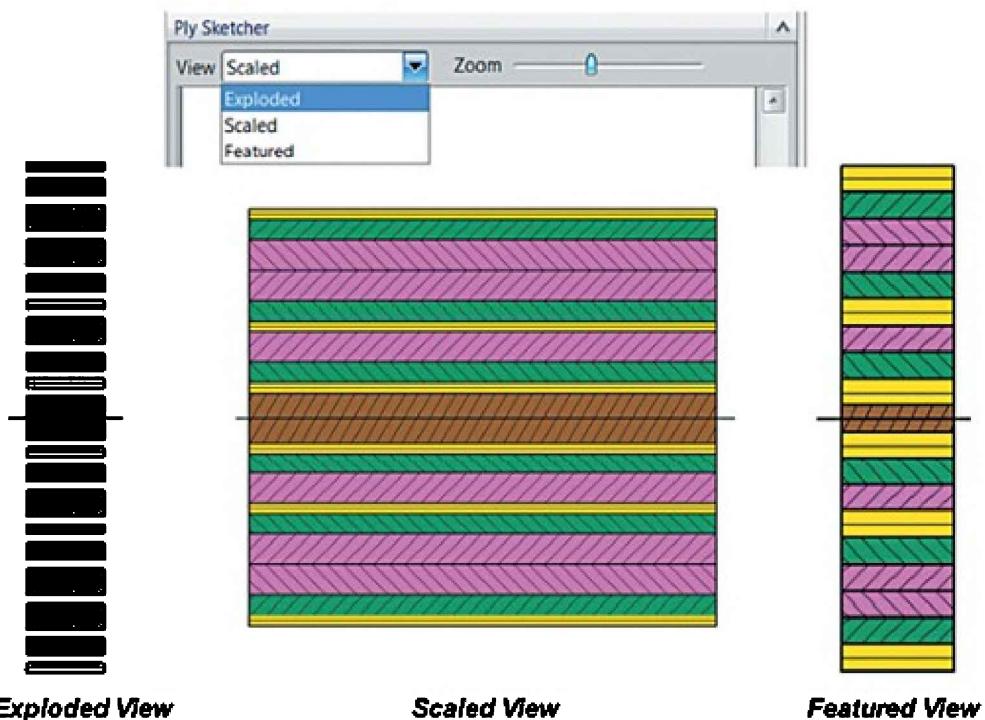


Figure 6.10. Types of layup scheme visualizations

The *Validation* group

This group can be divided into two parts: property validation and composite sample strength evaluation.

The validation of composite properties involves calculating equivalent material properties: density, temperature coefficient of linear expansion, and stiffness matrices A , B , D , and S (para. 6.6). The relevant information is displayed in tabular or text format when you use the following commands:

- The *Laminate Information to Text Windows* command outputs

all data on the composite and its properties in an information window.

- The **Laminate Information to Spreadsheet** command outputs all data on the composite and its properties in an *Excel* spreadsheet (Figure 6.11).
- The **Laminate Information to CSV File** command outputs all data on the composite and its properties in a CSV (comma-separated values) file.

Validation

Optim **Laminate Information to Spreadsheet**

Enable Optimization

Stiffness Matrices A,B,D,S

A	B	C	D	E	F	G	H	I	J
151,281E+6	48,868E+6	47,422E+6			-165,118E+6	-48,654E+6	-47,422E+6		
48,868E+6	58,436E+6	47,422E+6	B =		-48,656E+6	-50,044E+6	-47,422E+6		
47,422E+6	47,422E+6	50,834E+6			-47,422E+6	-47,422E+6	-50,805E+6		
287,879E+6	60,415E+6	58,062E+6			1,819E+6	000,000E+0			
60,415E+6	69,605E+6	58,062E+6	B =		000,000E+0	2,025E+6			
58,062E+6	58,062E+6	63,214E+6							
Mass Density	1,433E-6 kg/mm ³								
Young's Modulus	9,431E+6	1,463E+6				mN/mm ² (kPa)			
Bending Young's Modulus	53,260E+6	6,530E+6				mN/mm ² (kPa)			
Poisson's Ratio			0,399	0,062		Unitless			
Bending Poisson's Ratio			0,394	0,048		Unitless			
Shear Modulus			1,272E+6		1,010E+6	1,125E+6 mN/mm ² (kPa)			
Bending Shear Modulus			5,743E+6			mN/mm ² (kPa)			
Thermal Expansion	1,779E-6	19,717E-6	-19,404E-6			1/C			
Thermal Conductivity	1,682E+12	1,682E+12	000,000E+0			microW/mm·C			

Figure 6.11. Spreadsheet. Basic composite info

You can evaluate the strength of the composite sample with a specified layup and failure criterion using the following commands:

- You can use the **Analyze Laminate Strength** command to evaluate the strength of the sample under *Loads per Unit Length* as specified in this command. The results are written into a spreadsheet, a CSV file, or as a stress and strain diagram along the thickness of the composite (Figure 6.12, A). The red dots correspond to the bottom of each ply, and the green dots correspond to top of each ply.
- You can use the **Plot Failure Envelopes** command to evaluate

the carrying capacity of the composite sample as a strength curve for the specified ply (Figure 6.12, B). This curve reflects the limit stress of the sample at which failure occurs depending on the selected failure criterion and allowable stress for the material.

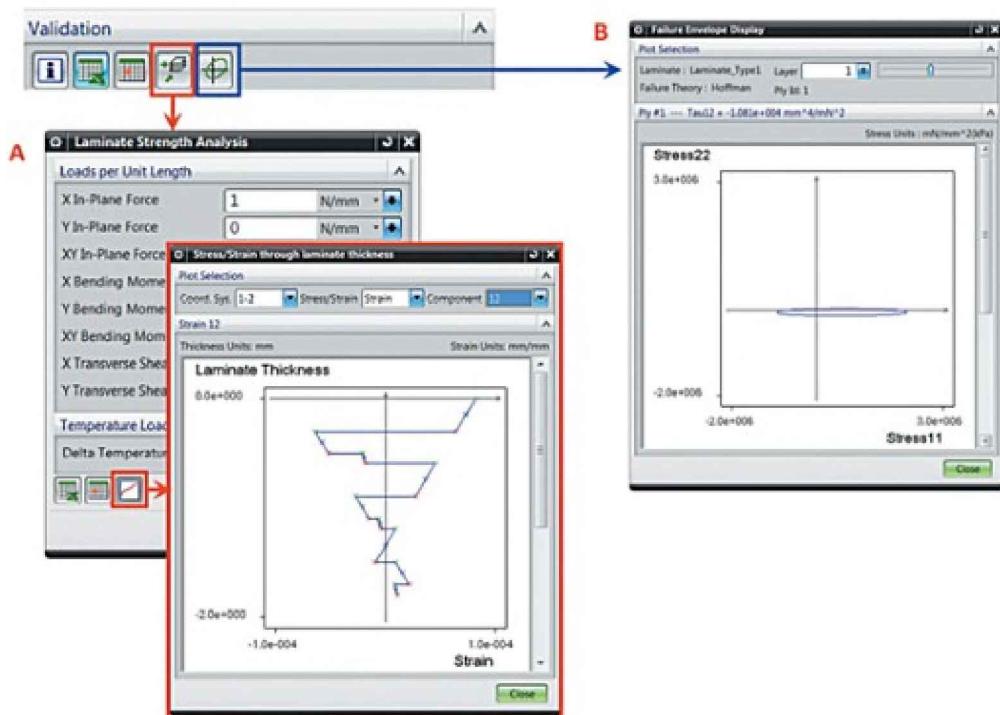


Figure 6.12. Evaluating the strength of a composite sample

The **Optimization** group

This group is a collection of tools for optimizing the composite sample. Five optimal layup variants are found by varying design parameters, objectives, and design constraints. The optimization method used in this application is based on a genetic algorithm. It is noteworthy that you can optimize both continuous and discrete parameters. You can also use predefined materials as discrete values of a variable parameter of the composite ply. Continuous variables used in this optimization can be used as variables for optimizing the entire structure.

The objectives can be mechanical or physical properties of the system, such as the value of the first natural frequency, the mass of the product, equivalent properties of the composite.

6.3. Creating composites using the global layup

The **Global Layup** method is based on defining composite properties by creating global plies on polygon geometry and specifying a draping method.

6.3.1. Creation and draping

As with the first method, you need to specify physical property tables for the composite material of the relevant FE meshes. To do so, in the FE mesh collector shortcut menu, select **Edit**. In the **Mesh Collector** dialog box, in the **Type** list, select **Laminate**. In the **Shell Property** box, select existing composite material properties from the list or specify new properties (**Create Physical...**). The **Laminate Modeler** dialog box appears.

In the **Laminate Properties** group, make sure that in the **Stacking Recipe**, you select *Inherited from Layup*. When you use the global layup, you should also check the **Zone Angle Tolerance** parameter responsible for the precision of creating zones with different physical properties. Leave the rest of the options the same as for physical property-based composite creation method (para. 6.2).

To further define the composite, use the **Global Layup** command on the **Laminates toolbar**. You can create global plies in two steps (Figure 6.13, A):

1. Use the tools in the **Layup Definition** group to create a layer stack. This step is completely the same as in the first composite creation method (refer to para. 6.2.2).
2. Use the **Define Draping Input** command to assign a set of faces and a draping method to the selected ply.

After you complete these steps, layup and global ply nodes are created in the model tree in the **Simulation Navigator** tab (Figure 6.13, B).

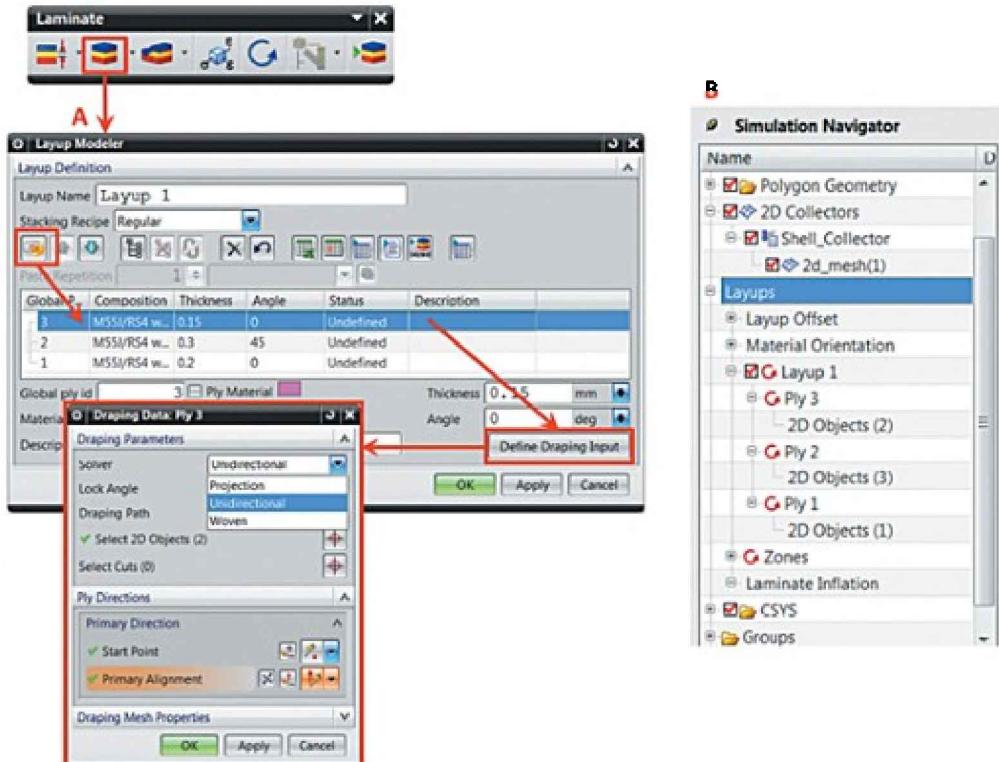


Figure 6.13. Creating and draping global plies

The second step needs a more detailed description. In the **Draping Data: Ply N** dialog box, perform the following actions:

1. In the *Solver* list, select the draping method:
 - Select *Projection* for plies of isotropic material, or when the layer is draped without reorienting fibers.
 - Select *Unidirectional* to start draping fibers in the specified direction. The draping is calculated with curvilinearity of surfaces taken into account.
 - Select *Woven* material to start draping warp fibers by specifying the *Primary Direction*, and weft fibers by specifying the *Secondary Direction*. The draping is calculated with fiber reorientation due to surface curvilinearity taken into account.
2. In the *Lock Angle* list, enter the critical distortion value for the angle between the warp and weft fibers during draping.
3. In the *Draping Path* list, select the method used to determine the draping direction:
 - Select *Geodesic* to drape along a vector.

- Select Seed Curve to drape using an existing curve.
- Under *Select 2D Objects*, select polygon faces to which the global ply is applied in the graphics area.
 - To reduce or eliminate the distortion of fiber stacking on surfaces for which a flat pattern cannot be constructed, you can use the so-called copes with the *Select Cuts* option.
 - In the *Ply Direction* group, specify fiber directions depending on the selected draping type. By default, warp and weft fibers are oriented at right angle.
 - In the *Draping Mesh Properties* group, specify the characteristic dimension that corresponds to the distance between a pair of nearest fibers. By default, the characteristic dimension of the draping mesh is measured in fractions of finite element size on the faces in question and is equal to 0.5. Select the *Specify Element Size* check box to specify the mesh dimension in units of length.

Figure 6.14 shows the draping of a woven fabric with coarse mesh (A) and medium mesh (B).

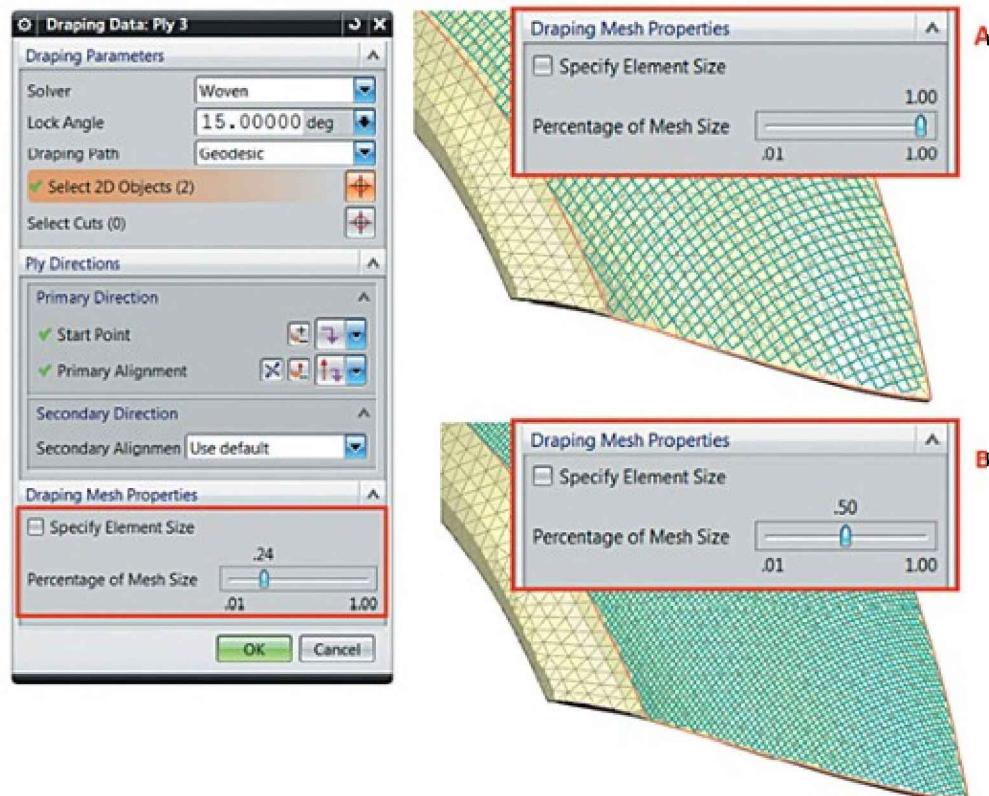


Figure 6.14. Draping a woven fabric

Note that draping is necessary for each global ply created in the **Global Layup** dialog box.

After the global plies are created and draped, in the **Simulation Navigator**, a *Layups* node appears, containing the following (Figure 6.13, B):

- *Layup Offset*.
- *Material Orientation*.
- *Layup N* and the plies as such: *Ply N*.
- *Zones* that are filled with content after a calculation run.

6.3.2. Layup offset and material orientation

When creating composites, it is important to maintain their offset from element faces, and the initial orientation of the material.

You can use *Layup Offset* to offset the layup in relation to a reference plane. The offset can be different for each polygon plane. To specify an offset, right-click each of the *Top*, *Middle*, or *Bottom* model tree nodes, use the **Edit** command, and select polygon faces. The number of assigned faces is shown in parentheses for each offset node, for example, *Middle(5)* means that the midsurface of the composite is matched with corresponding 2D finite element faces on five faces.

To specify a different offset value, use the shortcut menu of the *Layup Offset* to select the **Create User Defined Layup Offset Rule** command. In the dialog box, enter the offset from selected faces in the *Bottom Fiber Distance* box.

Note that you can assign only one offset condition to each face. If necessary, you can specify additional offset in mesh associated data: *Shell Offset*.

To verify the offset, you can display the 2D FE mesh thickness using the **Edit Display** command with the *Display 2D Element Thickness and Offset* check box selected.

The material orientation determines the zero angle of the composite ply orientation. You can specify material orientation in the mesh associated data

(refer to para. 6.2.1) by specifying the direction of the first axis or by selecting *Inherited from Layup*. In the latter case, layup fiber orientation is assigned using the *Material Orientation* model tree node:

- By default, the material orientation of the first ply is used, but you can double-click the *Default: First Ply* node to use the last ply instead.
- To use the material orientation of an intermediate ply, click *Material Orientation* to use the **New Orientation** command to select faces on which you need to use the material orientation of the specified ply.

6.3.3. Creating zones and additional tools for working with composites

Creating the so-called zones is a necessary stage of modeling composite materials. A zone is a selection of elements, which:

- Correspond to the same polygon face
- Belong to the same global layup
- Have identical orientation for each ply, give or take a specified tolerance

To create zones, you can use either the **Update Global Layups and Zones** command on the **Laminates** toolbar or the **Compute Zones** command from the shortcut menu of the **Zones** node (Figure 6.15, A). Stiffness matrices for calculations in NX Nastran are generated from the properties of composite zones.

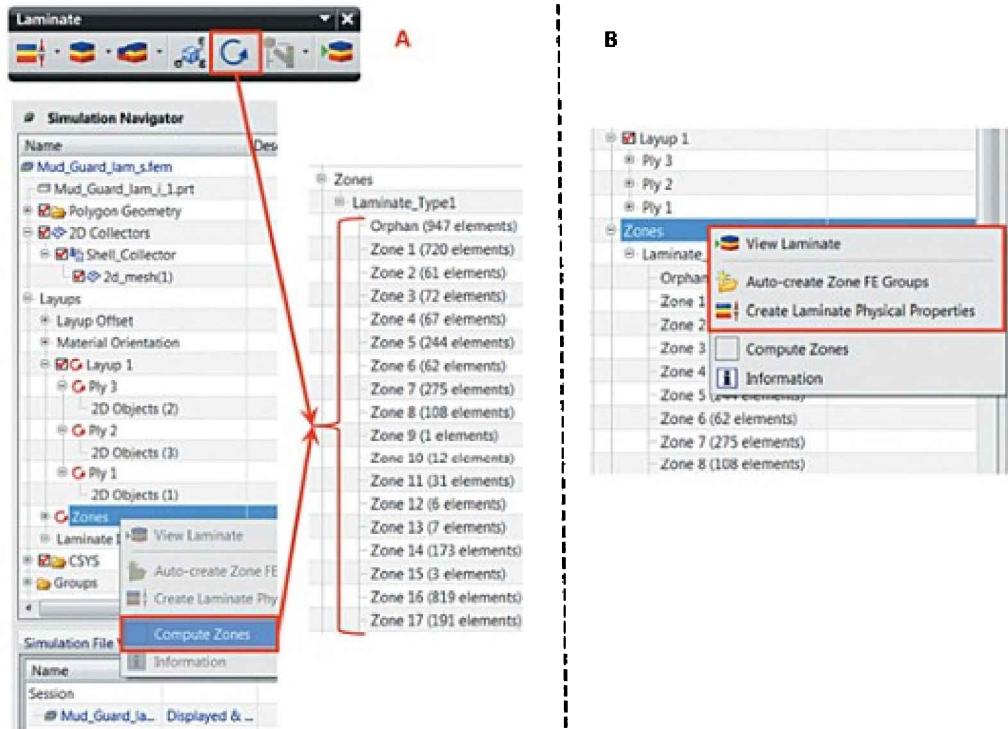


Figure 6.15. Creating and processing zones

The command opens the **Information** window that displays information about the number of created zones and describes errors that occurred during the zone computation. Each zone with its own unique physical properties is displayed in the model tree in the **Simulation Navigator**.

You can right-click a zone to use the following additional commands (Figure 6.15, B):

- Choose **View Composite** to display information on plies, their thicknesses and orientation on a *Ply Sketcher graphics representation*.
- Choose **Auto-create Zone FE Groups** to create a new group from this zone's elements. This group is added to the *Groups* node.
- Choose **Create Laminate Physical Properties** to create a set of *Physical Properties* that you can use to create other composites using the first method (para. 6.2) or simply analyze for strength in the **Laminate Modeler dialog box**. Note that physical property sets created in this way are not used in the model, so any modifications do not affect the created composite.

- Choose **Zone Properties** to display an information window with all layup parameters of the zone.

You can use additional commands to work with the layup and the global plies themselves. To use these commands, right-click the relevant node (Figure 6.16):

- Choose **Export Plies to FiberSim** to save the layup in a *1h* format file that you can import into FiberSim for detailed improvement.
- Choose **View Red Fibers** to use red color in the graphics area to highlight the layup distortion zone in accordance with the *lock angle* value.
- Choose **List Draping Orientation Options** to write draping results of the entire layup or a single ply to a spreadsheet or CSV file by specifying output objects: *Shear Angle*, *Yarn Angle*, *Primary Direction*, *Secondary Direction*, *Normal Direction*.
- Choose **Information** to open the **Information** window with information on materials, initial orientation and thickness of each ply of the layup.
- Choose **Auto-create Layup FE Group** to create a group of elements for the selected layup.
- Choose **Auto-create Ply FE Groups** to create a group of elements for the selected ply.
- Select **View Fiber Orientation** to display material orientation for elements of the selected ply. In critical zones determined in accordance with the lock angle value, the orientation direction is highlighted in red.
- Choose **View Flat Patterns** for a layup or **View Flat Pattern** for a single ply to view flat patterns as a graphical representation or in geometry format (Figure 6.16).

In addition to the previously described commands, you can use standard copy, rename, and delete commands on layups.

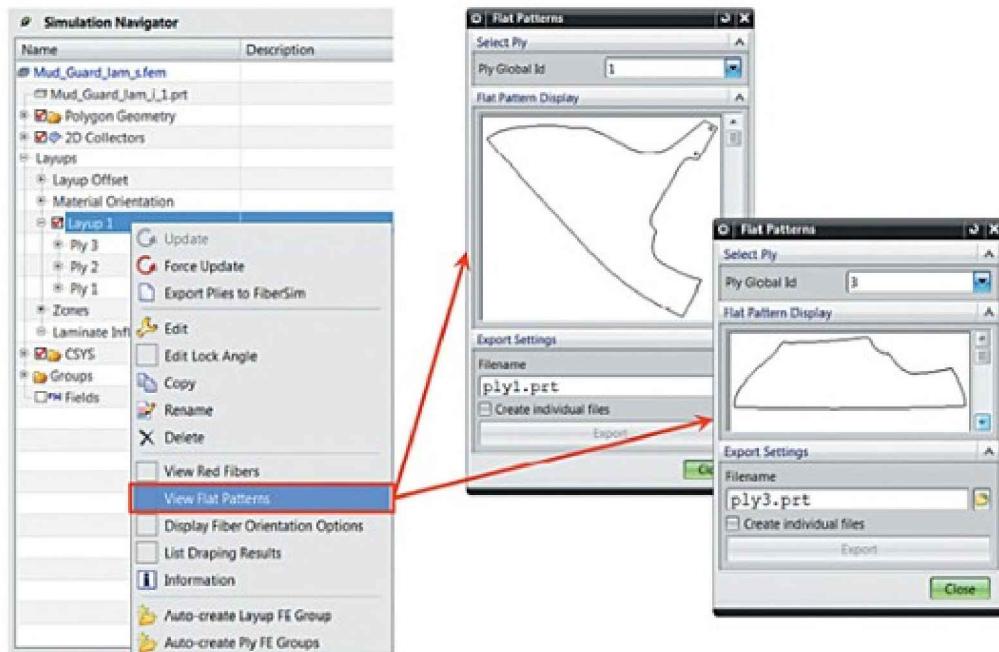


Figure 6.16. Creating flat patterns for each ply

6.3.4 Creating 3D composites

Instead of modeling composites using 2D finite elements, you can use 3D finite elements with physical properties of the composite retained. If you have a global ply layup, to create 3D FE models based on an existing 2D FE model, you can use commands of the **Laminates** toolbar:

- Use the **Extrude Laminate** command to select all polygon faces in the dialog box and have their 2D elements converted to 3D elements (Figure 6.17). To change the creation direction of the 3D elements, select the *Flip Element Normals* check box.
- Use the **Fill Laminate** command to create 3D elements between two face sets. You can only use this command if one of the face sets has a 2D mesh dependent on the other face set. In the dialog box, select the face set that has the independent mesh.

Note that you can only create 3D FE meshes using 2D finite elements with intermediate nodes.

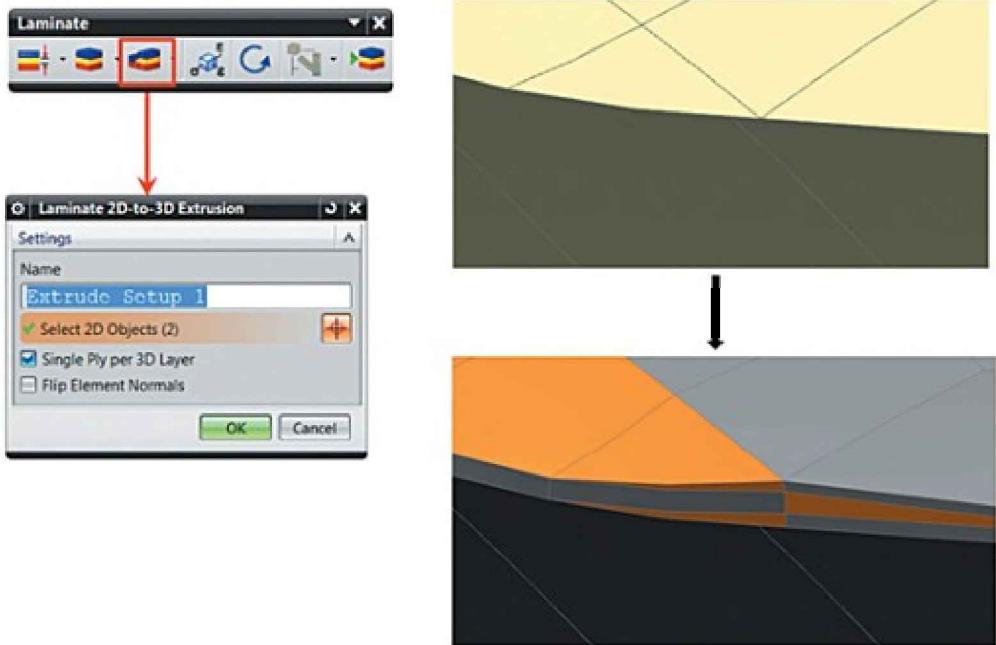


Figure 6.17. Creating 3D composites

The 3D element creation operations create a one layer-thick 3D finite-element layer for each ply of the composite based on global layup data. Depending on the particular command, the thickness of the resulting 3D FE model can be determined differently:

- If you use the ***Extrude Laminate*** command, the thickness of each 3D FE layer is the same as the thickness specified in the layup.
- If you use the ***Fill Laminate*** command, the total thickness of the 3D FE model is derived from the distances between the two face sets. The thickness of each ply is in this case proportional to the thickness specified in the composite layup.

In addition to the 3D finite elements, these commands cause the following:

- A ***3D Collectors*** node with the new 3D FE meshes is added to the model tree. The original 2D meshes automatically become auxiliary, that is, they no longer participate in the calculations.
- New physical property tables for 3D elements are created in accordance with the physical properties of the source 2D elements.
- 3D pyramid elements are created that are used for modeling the

transition between zones with different layup sequences. By default, physical properties of the binder are assigned to these elements.

To view and edit the new physical properties of 3D composites (Figure 6.18), use the **Edit** command in the 3D collector shortcut menu and open the **Edit** dialog box for physical properties of *Laminate N – Extrusion*. In the **Solid Laminate Modeler** dialog box, you can select ply and interlaminar failure criteria in the *Failure Theory* and *Inter-Laminar Failure Theory* options. Note that in addition to criteria available for the original 2D elements, there is the *Max Transverse Shear Stress* criterion, and for interlaminar strength analysis there are additional criteria:

- *Transverse Shear*
- *Normal Stress*

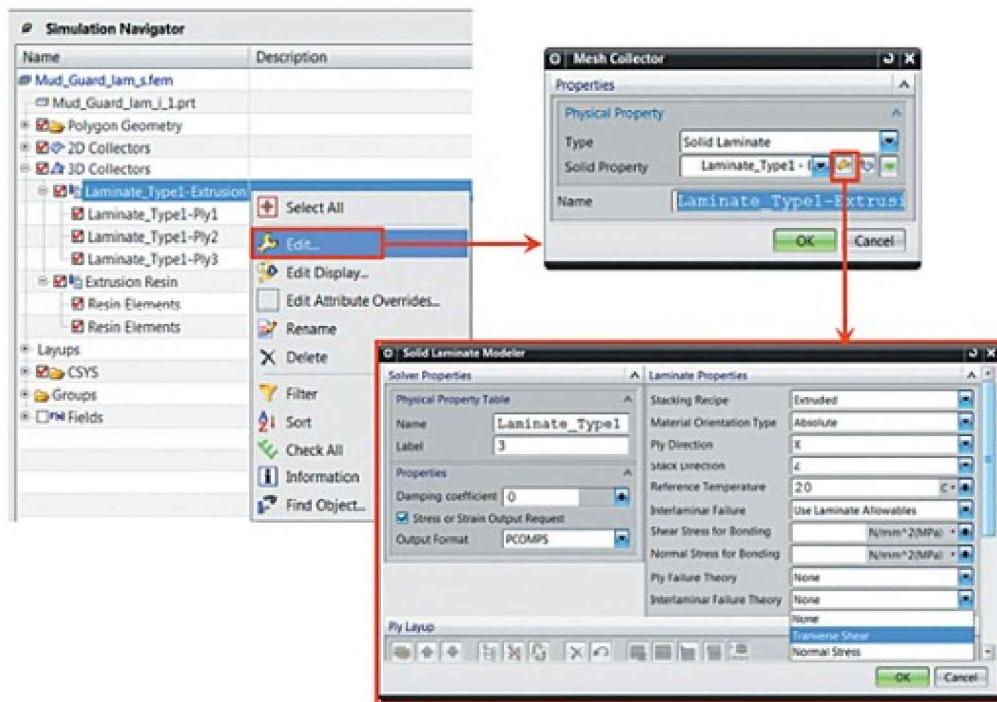


Figure 6.18. Editing 3D composite properties

6.4. Material types and microstructured materials

You can use the following materials for plies of composite structures:

- Isotropic material.
- Orthotropic Material.
- Ply material that is not a commonly recognized material and can be found only in composite plies. It combines the properties of fiber and matrix (binder) materials, which define its equivalent stiffness and strength properties.

6.4.1. Creating ply materials

You specify the **Ply Material** when you create the plies of the composite (Figure 6.19). Its properties are specified using either the **Ply Material** command on the **Laminates** toolbar or by directly setting it up when the material is specified for a ply.

After using the **Ply Material** command, in the **Laminate Ply Material Manager** dialog box, the **Ply Material List** table lists the ply materials that were created. To create a new ply material, select one of ply material types in the **Type** list:

- *Unidirectional*
- *Woven*
- *Particulate* (granular)
- *RandomShotFiber* (with randomly oriented short fibers)

Then open the **Composite Material Ply** dialog box by clicking the **Create** button and specify the following parameters (Figure 6.19):

- In the **Basic Information** group, specify the isotropic matrix material and its volume fraction, and the material of the other component (fiber or particulate) and its volume fraction. For woven type, enter the **Balance Coefficient** that defines the warp to weft fiber ratio and enter the **Weft Fiber Angle** to assign the angle between the warp and the weft fibers.
- When you specify material for plies that can have fibers with a particular cross-section, all types except *Particulate* support specifying the **Finished Thickness**.
- In the **Stress** and **Strain** tabs, the **Strength** option can be set to *Calculated*. In this case allowable stresses and strains are calculated based on user-defined allowed stresses and strains for

each of the components of heterogeneous ply material. If you set this option to *User defined*, you need to enter allowed stress or strain values of the equivalent ply material to evaluate the structure using failure criteria.

- In the *Tsai-Wu* and *Puck* groups you can enter the coefficients of the corresponding failure criteria.

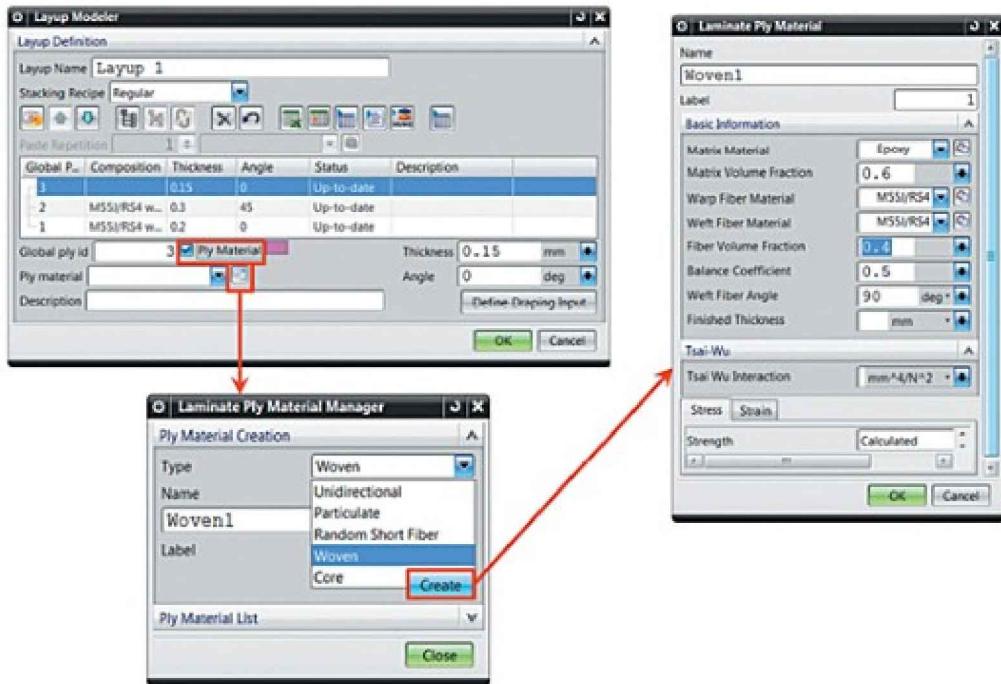


Figure 6.19. Specifying material properties for a microstructured ply

6.4.2. Micromechanics of ply materials

This paragraph describes each of the heterogeneous material types and the methods you can use to calculate effective properties of materials.

The following table lists material types (isotropic/orthotropic) that can be used to specify fiber and matrix materials, and types of material equivalent properties depending on the microstructure of the material.

Microstructure type	Fiber material type	Matrix material type	Equivalent material type
<i>Unidirectional</i>	Orthotropic		Isotropic or orthotropic
<i>Woven</i>	Orthotropic	Isotropic	Isotropic or orthotropic

<i>Particulate</i>	Isotropic	Isotropic
<i>RandomShotFiber</i>	Isotropic	Isotropic or orthotropic

Equivalent properties of ply material are determined based on material properties of its components. Different relations are used depending on the material microstructure type.

The following notation is used in the following text:

- The f superscript is used for *fiber* properties
- The m superscript is used for *matrix* properties

For all discussed material microstructure types, you need to enter volume fractions of fiber and matrix when you create the ply material:

- V^f : volume fraction of fiber ($0 < V^f < 1$)
- V^m : volume fraction of matrix ($0 < V^m < 1$)
- $V^f + V^m \leq 1$

The following basic relations are used to calculate equivalent material properties in the ply coordinate system (refer to para. 6.1.2)

Unidirectional fibrous material

Unidirectional fiber composites contain strong unidirectional fibers permeated with a relatively weak matrix.

Equivalent material properties are calculated as follows [19]:

Ply Young's modulus	Ply Poisson's ratio	Ply shear modulus
$E_1 = V^f E_1^f + V^m E^m$	$\nu_{12} = V^f \nu_{12}^f + V^m \nu^m$	$\frac{1}{G_{12}} = \frac{V^m}{G^m} + \frac{V^f}{G_{12}^f}$
$\frac{1}{E_2} = \frac{V^m}{E^m} + \frac{V^f}{E_2^f}$	$\nu_{13} = \nu_{12}$	$G_{13} = G_{12}$
$E_3 = E_2$	$\nu_{23} = \nu_m$	$G_{23} = G_m$
Ply density	Ply specific heat	
$\rho = V^f \rho^f + V^m \rho^m$	$c_p = \frac{1}{\rho} (V^f \rho^f c_p^f + V^m \rho^m c_p^m)$	
Ply thermal conductivity	Ply coefficient of thermal expansion	
$K_1 = V^f K_1^f + V^m K^m$	$\alpha_1 = \frac{(\alpha_1^f E_1^f - \alpha^m E^m) V^f + \alpha^m E^m}{(E_1^f - E^m) V^f + E^m}$	
$\zeta = \frac{1}{4 - 3\nu^m}$		
$\eta = \frac{\frac{K_1^f}{K^m} - 1}{\frac{K_1^f}{K^m} + \zeta}$	$\alpha_2 = \alpha^m + (\alpha_2^f - \alpha^m) V^f +$	
$K_2 = K^m \frac{1 + \zeta \eta V^f}{1 - \eta V^f}$	$\left(\frac{E_1^f \nu^m - E^m \nu_{12}^f}{E_1} \right) (\alpha^m - \alpha_1^f) V^m V^f$	
$K_3 = K_2$	$\alpha_3 = \alpha_2$	

Woven material

Woven fiber composites contain warp fibers and weft fibers at an angle to each other. These fibers are permeated with the matrix.

When you specify the material, you enter the *Balance Coefficient k* to define the ratio of warp fiber to weft fiber.

Equivalent material properties are calculated as follows:

1. The woven ply is represented as three layers (Figure 6.20): two outer unidirectional plies of thickness $k/2$ at angle 0° (warp fiber direction) and one inner unidirectional layer of thickness $(1 - k)$ at angle 90° (weft fiber direction).
2. For each new unidirectional layer, equivalent material properties are calculated as previously shown.
3. Using the calculated properties of each layer and layup angles, effective properties of the composite are determined for the 1–2 coordinate system (Figure 6.20), where 1 is the warp direction. These properties are calculated in a similar way to the calculation of composite effective properties in **Laminate Modeler**, including transverse shear moduli.

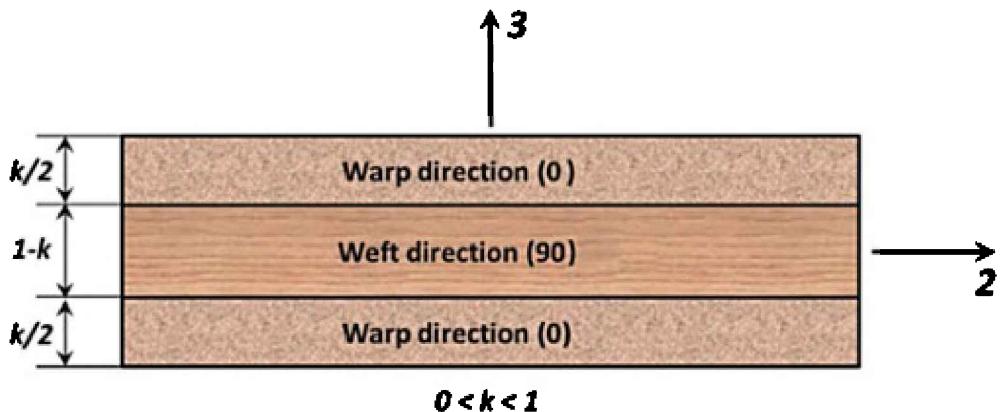


Figure 6.20. Representing a ply as a three-layer stack

The rest of the equivalent material properties are calculated as follows [20]:

Ply Young's modulus $E_3 = E_m$	Ply Poisson's ratio $\nu_{13} = k\nu_{12}^{\text{warp ply}} + (1 - k)\nu_m$ $\nu_{23} = (1 - k)\nu_{12}^{\text{warp ply}} + k\nu_m$
Ply thermal conductivity $\frac{1}{K_3} = \frac{k}{K_2^{\text{warp ply}}} + \frac{1 - k}{K_2^{\text{weft ply}}}$	Ply coefficient of thermal expansion $\alpha_3 = k\alpha_2^{\text{warp ply}} + (1 - k)\alpha_2^{\text{weft ply}}$

Particulate material

Particulate materials contain solid particles suffused in matrix.

When you calculate equivalent properties of a material, all components are presumed isotropic, and the properties of the resulting material are isotropic as well.

Equivalent material properties are calculated as follows [21]:

Ply Young's modulus	Ply Poisson's ratio
$E^{pc} = \frac{(V^f)^{2/3} E^m}{1 - (V^f)^{1/3} (1 - E^m/E^p)} + \left(1 - (V^f)^{2/3}\right) E^m$	$\nu^{pc} = \frac{E^{pc} - 2G^{pc}}{2G^{pc}}$
Ply shear modulus	Ply density
$G^{pc} = \frac{(V^f)^{2/3} G^m}{1 - (V^f)^{1/3} (1 - G^m/G^p)} + \left(1 - (V^f)^{2/3}\right) G^m$	$\rho = V^f \rho^f + V^m \rho^m$
Ply thermal conductivity	
$K^{pc} = \frac{(V^f)^{2/3} K^m}{1 - (V^f)^{1/3} (1 - K^m/K^p)} + \left(1 - (V^f)^{2/3}\right) K^m$	
Ply coefficient of thermal expansion	
$\tilde{\alpha} = \alpha^m - (V^f)^{1/3}(\alpha^m - \alpha^p)$	
Ply specific heat	
$c_p = \frac{1}{\rho} \left(V^f \rho^f c_p^f + V^m \rho^m c_p^m \right)$	

where the f superscript denotes particle properties, the m superscript denotes binder properties, and the pc index denotes particulate material properties.

Randomly-distributed fiber material (RandomShortFiber)

RandomShortFiber materials consist of randomly oriented short fibers permeated with a matrix. The fibers are presumed to be longer than the ply is thick, and they are presumed to lie in the plane of the ply. The same approach is applied as for woven materials [20] but the ply is represented by a balanced stack of unidirectional layers: layers with 0° and 90° angles, and other layers with angles varying uniformly from 0° through 90° . The number of these layers depends on whether the condition shown on Figure 6.21 is fulfilled, but there can be at most 512 layers.

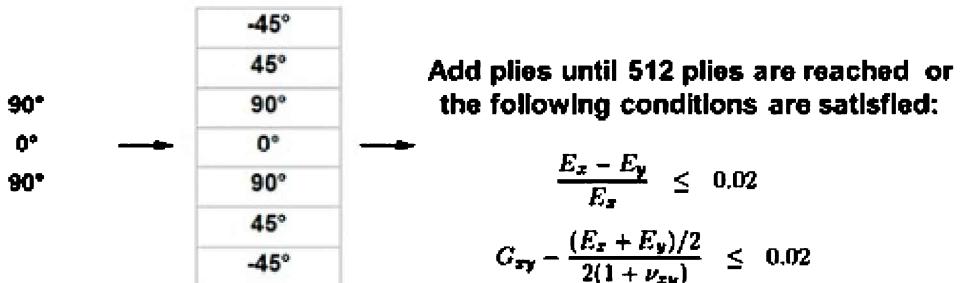


Figure 6.21. Representing a ply as a multiple-layer stack

6.5. Solution parameters and viewing results

6.5.1. Solution parameters

Before you analyze a structure containing composite materials, you need to check solution parameters in the **Solver Properties** group of the **Laminate Modeler** dialog box (refer to para. 6.2) of the physical property table of the corresponding FE mesh. The following is a description of key parameters that determine special aspects of solution and result output for composite materials.

- *Output Format*: select the output format of the composite properties record (card) (*PCOMP*, *PCOMPG*, *PSHELL*, or *PCOMPS*) that is used when exporting the FE model to an NX Nastran *input file*. Each of the format has its own aspects, for example, you can use *PCOMPG* to display results for global plies, *PSHELL* only supports the output of resultant results of the shell (results per ply are not available), *PCOMPS* is responsible for 3D composite properties.
- *Nonstructural Mass*: add mass value per unit of area of the element set in question only for *PCOMP*, *PCOMPG*, and *PSHELL* cards.
- *Damping Coefficient*: enter and use the damping coefficient of the material (structural or hysteretic damping) for *PCOMP*, *PCOMPG*, and *PCOMPG*.
- *Stress or Strain Output Request*: if this check box is selected, results are also written into a **f06 text file*.
- *Laminate Options*: manage recording of particular stiffness properties that define composite behavior for *PCOMP* and

PCOMPG cards:

None: full set of properties.

MEMBrane: membrane properties only.

BENDing: bending properties only.

SMEAR: ignore ply layup without shear and bending-membrane properties.

SMCORE: similar to *SMEAR* but used only for symmetric layup with filler. Not used with *PCOMPG* card.

SMCORE+transverse shear: similar to *SMCORE* but takes shear properties into account. Used only with *PSHELL* card.

6.5.2. Special aspects of results visualization

The type and availability of data recorded in the results file for the solution depend on options in the *Solver Properties* group of the **Laminate Modeler** properties (para. 6.5.1).

The following table matches output formats with values that are written into the results file.

Value	Output format	Result type
Force	PCOMP, PCOMPG, PSHELL	Resultant forces of shell elements
	PCOMPS	Resultant forces of solid-body elements
Stress	PCOMP PCOMPG	Stresses in plies Failure indices and strength coefficients in plies Interlaminar failure indices and strength coefficients
	PCOMP, PCOMPG, PSHELL	Stresses in shell elements
	PCOMPS	Stresses in plies Failure indices and strength coefficients in plies

Strain	PCOMP PCOMPG	Strain in plies Failure indices and strength coefficients in plies Interlaminar failure indices and strength coefficients
	PCOMP, PCOMPG, PSHELL	Strain in shell elements
	PCOMPS	Strain in plies Failure indices and strength coefficients in plies

To view and analyze results, go to the **Post Processing Navigator** tab of the Resource Bar and load the results file. In addition to standard results viewing tools (Figure 6.22, A) described in the previous chapter, you can use the so-called **Advanced Post Report** for detailed analysis of behavior of composite structures and strength evaluation.

You can start this command on the **Post Processing** toolbar. In the dialog box, select the required solution, step, and iteration. When you click **OK** or **Apply**, in the **Post Processing Navigator**, a *Laminate Post Report N* node is created. To generate results for the selected solution, right-click the results node in the **Simulation Navigator** tab of the simulation model. The shortcut menu contains the following commands (Figure 6.22, B):

- **Create Spreadsheet Report**
- **Create Graphical Report**
- **Create Quick Report**

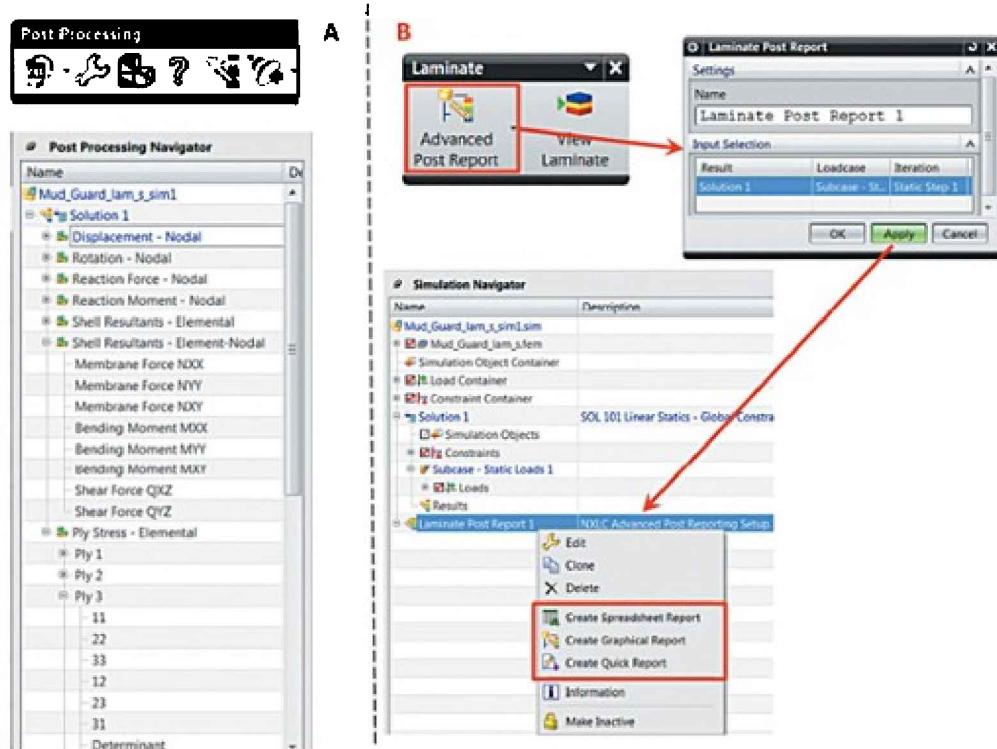


Figure 6.22. Results visualization commands for composites

These commands create new nodes in the **Simulation Navigator** simulation model tree. To view results in graphical form, as a spreadsheet, or as text, right-click the relevant node and choose the corresponding command.

These commands use only composite properties of the FE model, so you can edit the failure criterion or even change composite properties to obtain new results without recomputing the problem.

The **Create Spreadsheet Report** command creates a report based on the calculation results as an *Excel* spreadsheet or text file that contains information on stresses and strain in plies, failure indices, and strength coefficients (Figure 6.23, A). You can also use it to:

- Create a report for individual stress and strain components for plies in the composite or ply coordinate system.
- Output information on maximum and minimum values.
- Set up output filtering using results, elements, or plies.
- Set up result sorting for output.
- Create a report for resultant results of the shell.

The report you create corresponds to a new node of the simulation model tree that can be exported to *Excel* and *CSV* files.

You can use the **Create Graphical Report** command to calculate envelopes for values based on one or more results. You can view these in the **Post Processing Navigator** tab (Figure 6.23, B):

- Stress and strain in plies
- *Failure Index*
- *Strength Ration*
- *Margin of Safety*, factor of safety

All these coefficients are interchangeable and can be expressed in terms of one another. They differ in the range of safe values. For example, all elements with a failure index value greater than 1 indicate a possibility of failure in the areas.

To obtain result envelopes after using the **Create Graphical Report** command, right-click the new *Graphical Report N* node and choose **Generate Result File**. A binary results file is created, containing the worst results among selected solutions, steps, iterations, and plies.

Note that these results correspond only to the midsurface of the ply.

You can use the **Create Quick Report** tool to output the data in *Excel* spreadsheet or *CSV* text file formats. These data can contain information on maximum and minimum values and plies, elements, and nodes where these values occur (Figure 6.23, C).

An important advantage of this command is that you only need a results file to use it, therefore, you do not need to load an FE model.

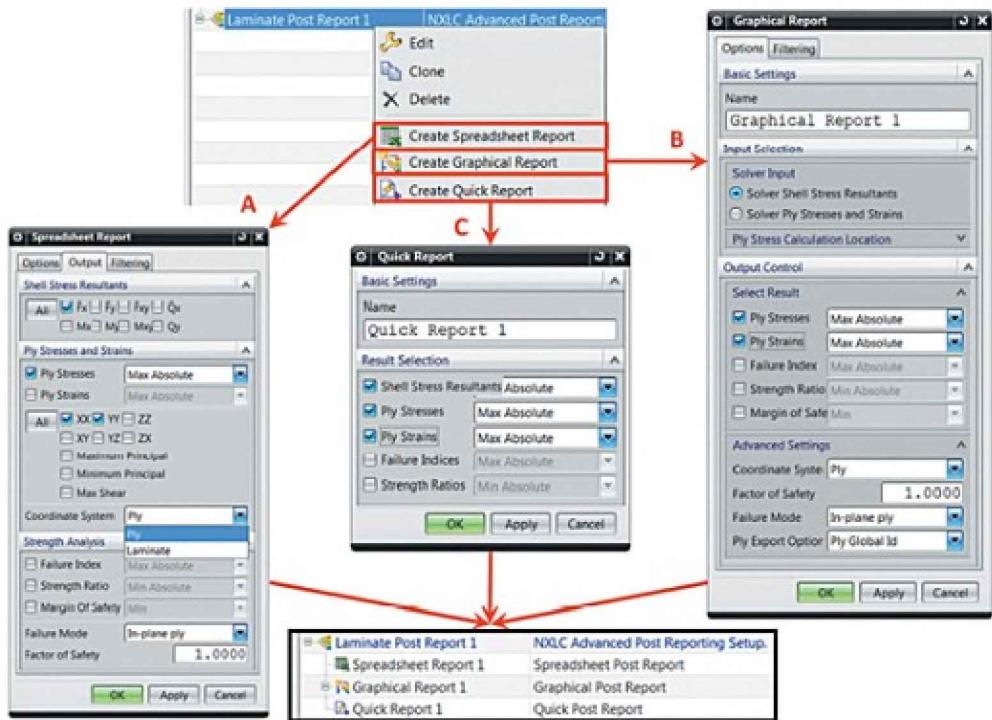


Figure 6.23. Features of the Advanced Post Report command

6.6. Summary of composite theory

The operation of NX Laminate Composites is based on the *First-order Shear Deformation Theory (FSDT)* [22]. This theory is similar to the *Classical Lamination Theory (CLT)*, differing mostly in the following aspects:

- It is assumed that *CLT* does not have shear strain, whereas *FSDT* calculates shear strain and forces along the thickness of the composite.
- *FSDT* requires continuity of transverse displacement and rotational degrees of freedom in the plane. On the other hand, *CLT* requires continuity of transverse displacement and its first-order derivative over two coordinates of the plane.

This is why *FSDT* is well-suited to finite-element discretization.

Here is a list of *FSDT* assumptions:

- Upon deformation, sections perpendicular to the reference plane remain plane

- Each ply is considered under assumption of flat-stressed condition
- The plies are ideally glued
- Deformations and displacements are small

Producing results in NX Lamine Composites involves the following steps:

1. Resolving the defining relations of plies
2. Converting relations of plies from the ply coordinate system to the composite coordinate system
3. Computing stiffness matrices for the composite
4. Computing curvature and strain at midplane for the composite based on resultant shell stresses
5. Computing strain for each ply
6. Computing stresses for each ply

You can find a detailed description of the *FSDT* theory and all relevant relations in [22]. Only stiffness matrices of the composite are described in this book.

The composite stiffness matrix $[ABD]$ (6x6) is used to compute strain from resultant shell stresses. It is of the following form:

$$[ABD] = \begin{bmatrix} [A] & [B] \\ [B] & [D] \end{bmatrix}$$

$$A_{ij} = \sum_{k=1}^N \bar{Q}_{ij}^{(k)} (z_k - z_{k-1})$$

$$B_{ij} = \frac{1}{2} \sum_{k=1}^N \bar{Q}_{ij}^{(k)} (z_k^2 - z_{k-1}^2)$$

$$D_{ij} = \frac{1}{3} \sum_{k=1}^N \bar{Q}_{ij}^{(k)} (z_k^3 - z_{k-1}^3)$$

where

$[A]$ is the membrane stiffness matrix

$[B]$ is the bending membrane stiffness matrix

$[D]$ is the bending stiffness matrix

i, j are axes of the ply coordinate system

N is the total number of plies in the composite

k is the number of the ply

z corresponds to the position of the ply with regard to the reference plane

The transverse shear matrix $[S]$ (2x2) defines the correspondence of composite strain energy to the strain energy associated with the distribution of shear stress caused by bending and shear in the XY plane of the composite coordinate system.

6.7. Example. Static analysis of a composite product

In this example, a prepared simulation FE model of a corrugated panel (Figure 6.24) is used to demonstrate setting up composite material parameters, performing a static analysis, and analyzing the obtained results.

In order to demonstrate NX Laminate Composite features, the composite is defined using two methods to create composite properties of a ply and verify strength parameters of the composite structure created.

The proposed problem can be solved by performing a number of steps:

1. Open the FE model and simulation model files
2. Set up the composite material using the **Laminate Modeler** tool
3. Drape the composite
4. Perform the static analysis
5. View static analysis results for the composite product

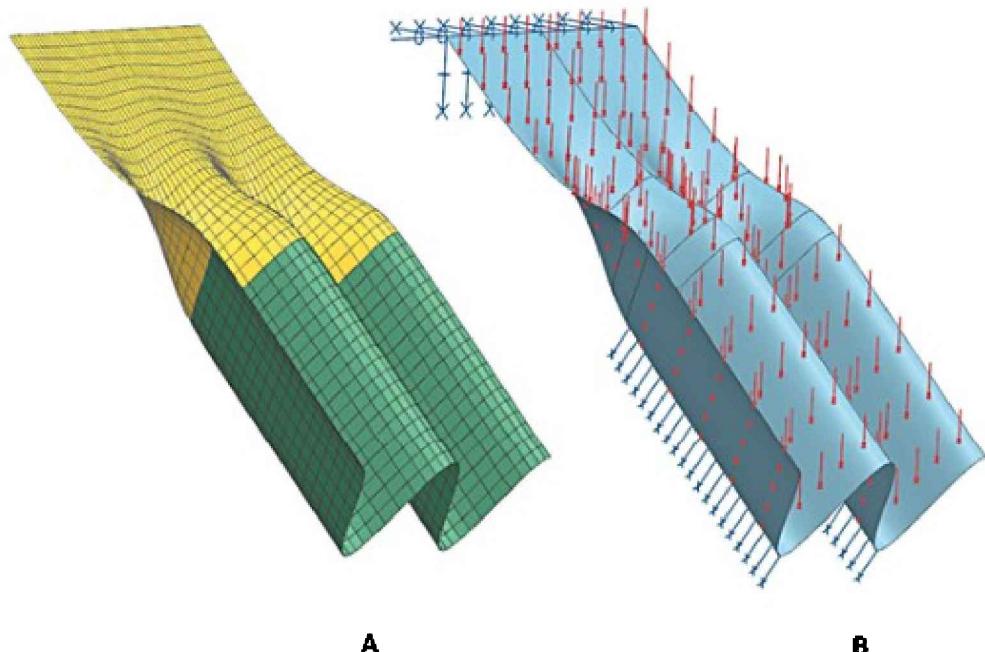


Figure 6.24. Roofing structure element A. FE model. B. Boundary conditions.

The original FE model is a set of shell elements that describe the thin-shell

part of the structure. The structure is under pressure, the top edge of the structure has a symmetry condition set up, and at the bottom, two edges have supported constraint set up.

6.7.1. Opening the FE model and simulation model files

Launch NX and open the *Gofra_sim.sim* file.

To reset properties of the dialog boxes to default, choose the following command in the main menu: **Preferences→User Interface→General→Reset Dialog Box Settings**, click **OK**.

In the **Simulation File View** dialog box, the simulation model file is active. Double-click the FE model file to switch to it.

Note that the model tree of the **Simulation Navigator** includes two physical property sets in the **2D Collectors** node: **Comp_Modeler** and **Layup_Modeler**.

6.7.2. Setting up the composite material using the Laminate Modeler tool

In the **Simulation Navigator** tree view, right-click the **Comp_Modeler** 2D collector and choose **Edit**.

In the **Mesh Collector** dialog box, in the **Type** list, select **Laminate** (Figure 6.25), then click **Create Physical...** next to **Shell Property**.

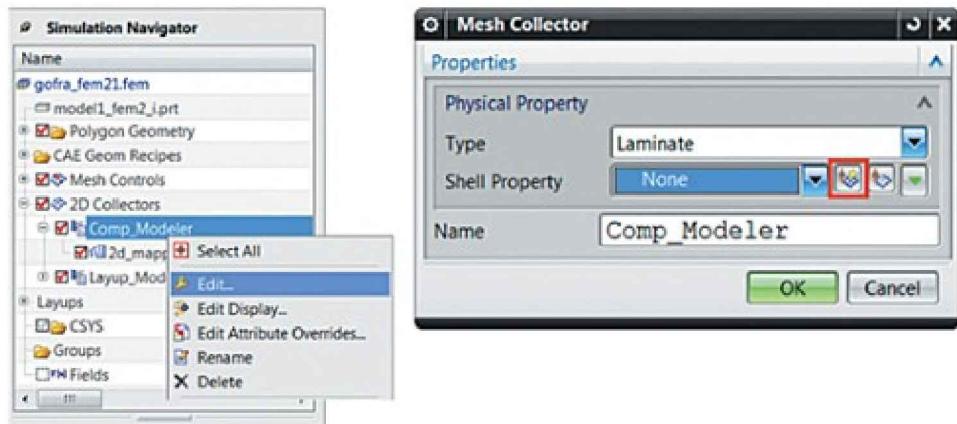


Figure 6.25. Mesh Collector dialog box

In the *Ply Layup* group, use the **Import Layup Using Shorthand Format** command to set up a sequence of four plies oriented at 45 degrees with regard to each other (Figure 6.26):

- Enter **[0/45/90/135]**
- Enter the *Thickness* value of **1 mm**
- Select **M55J/RS4 weave** in the *Material* list
- Click **OK**.

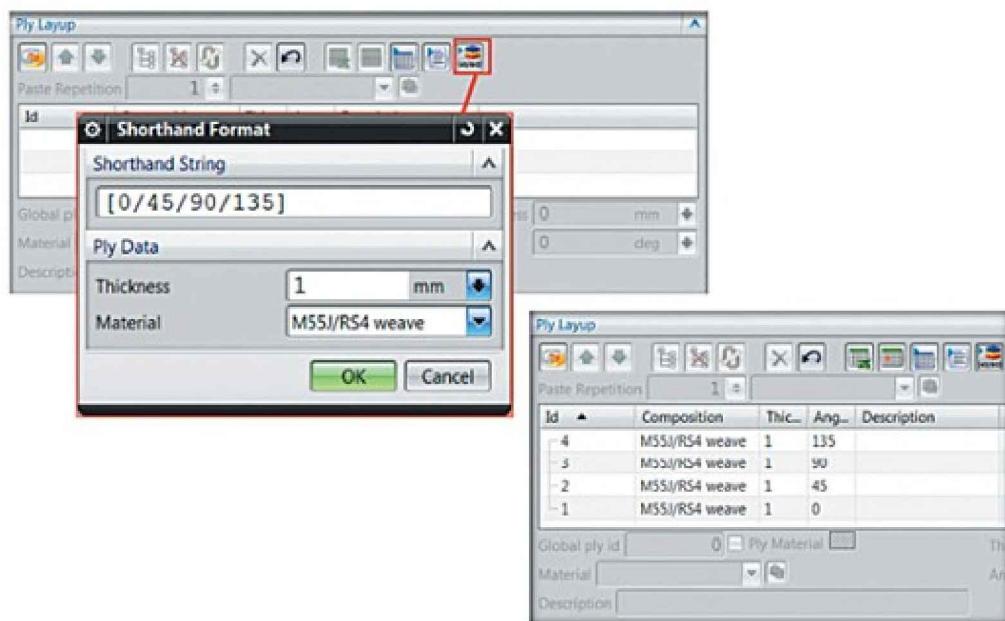


Figure 6.26. Quick layup creation dialog box

Hold **CTRL** and select the second and fourth plies in the table. Select the *Ply Material* check box. Next to *Ply Material*, click **Select Ply Material** (Figure 6.27).

In the **Laminate Ply Material Manager** dialog box, in the *Type* list, select **Unidirectional**, then click **Create**.

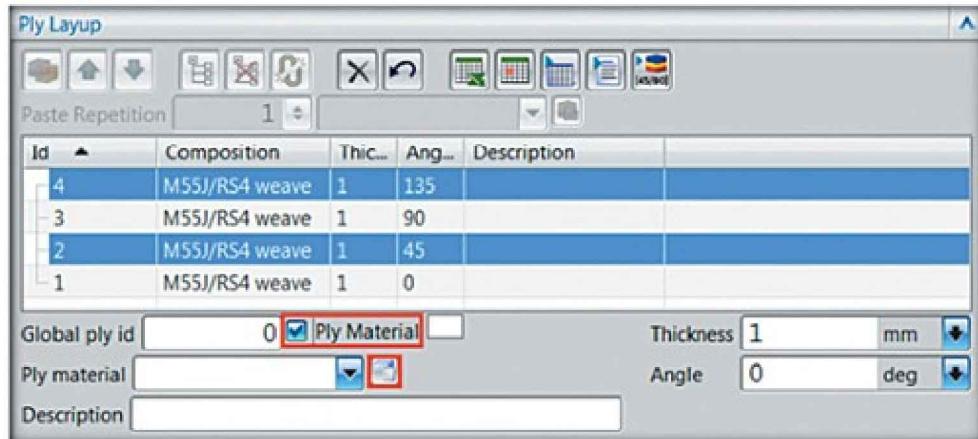


Figure 6.27. Specifying the ply material

In the **Composite Material Ply** dialog box (Figure 6.28), do the following:

- Click *Select Material* next to *Matrix Material*. Select *Epoxy* as the material
- Under *Matrix Volume Fracture*, enter *0.4*
- Click *Select Material* next to *Fiber Material*, select *E-glass* as the material
- Enter *0.5* in the *Fiber Volume Fracture* box
- Click **OK**

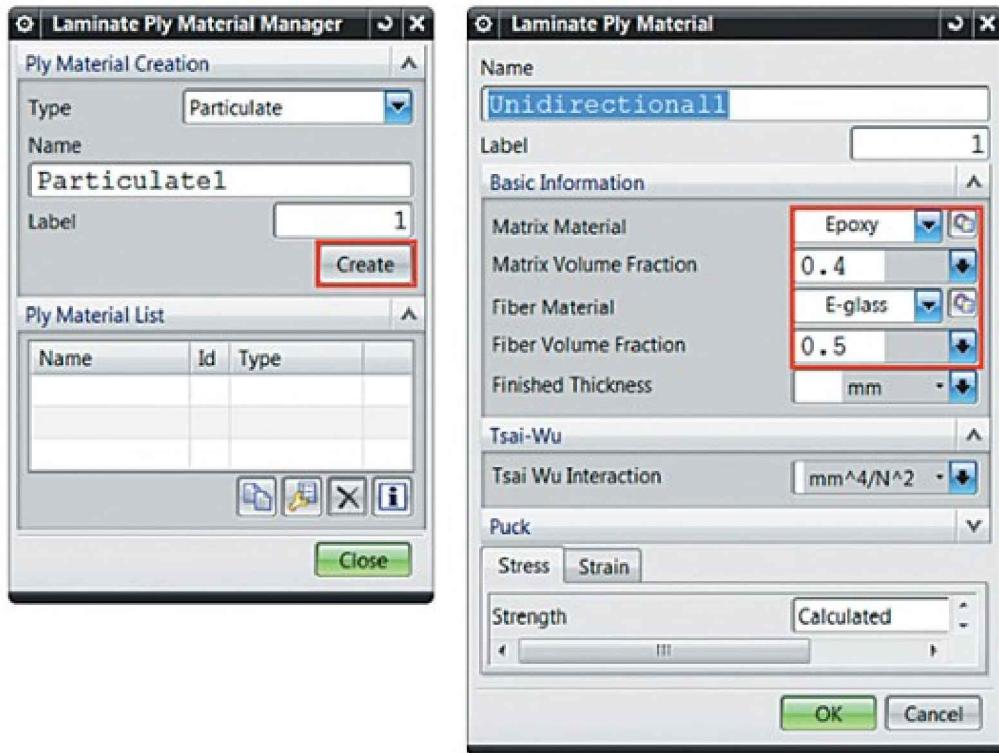


Figure 6.28. Specifying the composite ply material

Close the **Laminate Ply Material Manager** dialog box. Make sure to assign the **Unidirectional1. material to the second and the fourth plies.**

In the **Laminate Properties** group, in the **Failure Theory** list, select **Hoffman**, in the **Allowable Stress For Bonding** box, enter 1000 MPa.

In the **Validation** group, click **Analyze Laminate Strength**. In the **X In-Plane Force** and **Y In-Plane Force** dialog boxes, enter **5000 N/m** and **10000 N/m** respectively, then click **Export Spreadsheet**. In the **Excel** file, select the **Strength Analysis** tab, and find the **Allowable Loads on Laminate** data block. This block contains information on maximum allowable loads. Return to NX, close all dialog boxes.

Material properties for the part of the structure are now defined. However, since the properties of the composite material are orthotropic, you need to specify the orientation of the material. To do so, select all meshes of the relevant collector, right-click and choose the **Edit Mesh Associated Data** command in the shortcut menu. In the **Mesh Associated Data** dialog box, in the **Material Orientation Method** list, select **MC/D**. Click **OK**.

6.7.3. Draping the composite

In the **Simulation Navigator** model tree, right-click the *Layup_Modeler* 2D collector and choose **Edit**. In the **Mesh Collector** dialog box, in the **Type** list, select *Laminate* (Figure 6.25), then click **Create Physical....**

In the *Laminate Properties* group, in the *Failure Theory* list, select *Hoffman*. In the *Allowable Stress For Bonding* box, enter *1000 MPa*. In the *Failure Theory* list, select *Inherited from Layup*. Click **OK** in all dialog boxes. Use the **Global Layup** command on the **Laminate Modeler** toolbar. In the dialog box, click *Import Layup from a Laminate PPT*. In the new dialog box, select *Laminate1* from the list (Figure 6.29). Select all plies in the table and click **Define Draping Input**.

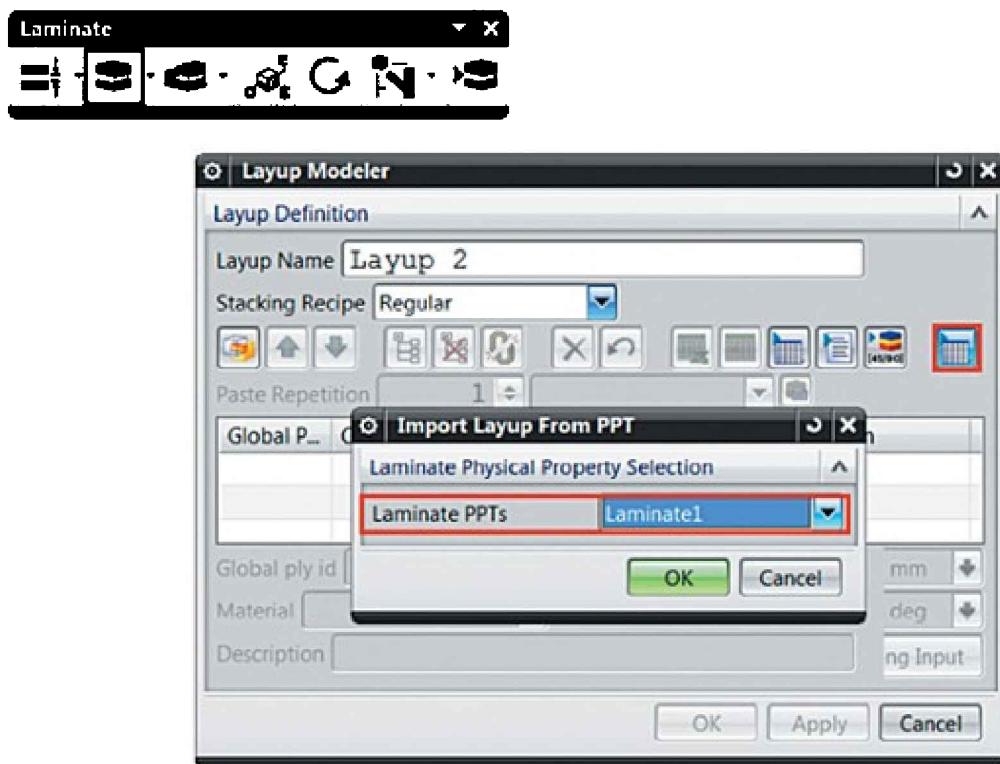


Figure 6.29. Importing the layup

In the dialog box, do the following (Figure 6.30):

- Make sure you select *Unidirectional* as the *Solver*
- Select five faces as 2D objects

- Set up copes by specifying 2 edges
- Specify an origin point and a vector coinciding with the Y axis
- Click **OK**.

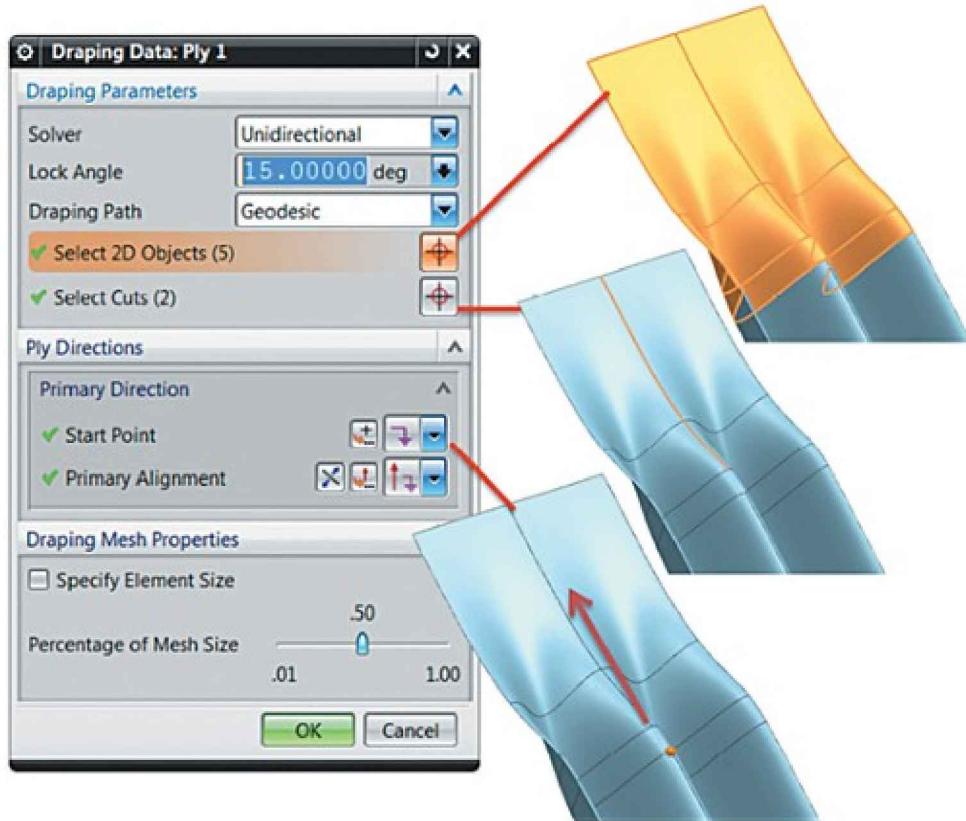


Figure 6.30. Creating a draping

In the table, select the third ply from the top and click **Create New Ply**. Specify Steel as the new ply material and enter a *Thickness* value of 0.5 mm. Click **Define Draping Input**.

In the dialog box, do the following (Figure 6.31):

- In the *Solver* list, select *Projection*
- Specify two faces as 2D objects
- Click **OK** in all dialog boxes

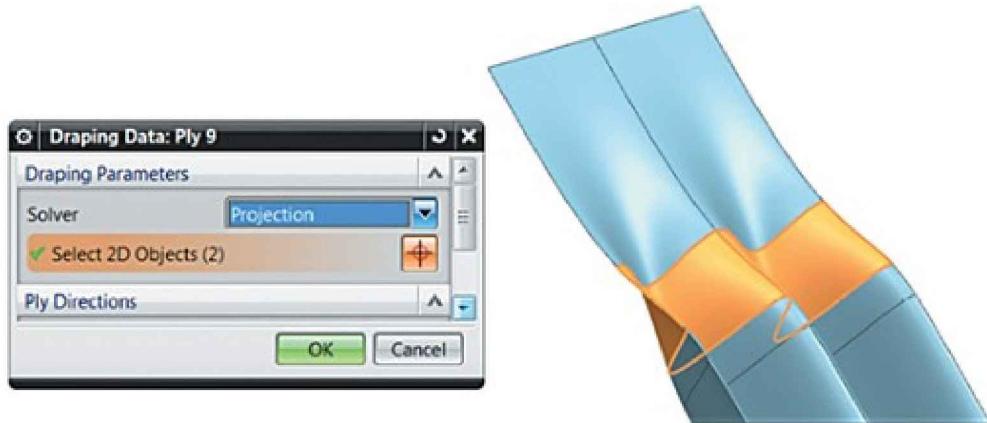


Figure 6.31. Creating a draping

To calculate the layup, use the **Update Global Layups and Zones** command of the **Laminates toolbar**.

The draping process is shown in the graphics area (Figure 6.32). Note the **Information** dialog box that contains information on the number of zones created and the layup quality at each ply.

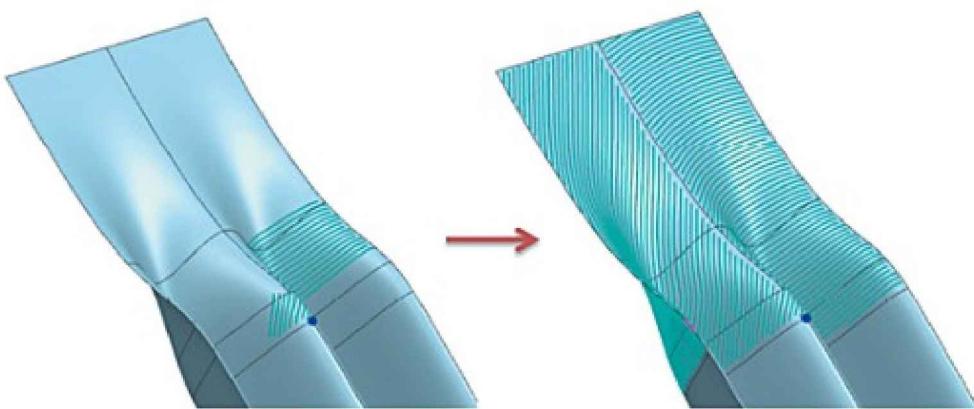


Figure 6.32. Creating zones

To specify the orientation of the material, select all FE meshes of the **Layup_Modeler** collector, right-click, and choose **Edit Mesh Associated Data**. In the dialog box, in the **Material Orientation Method** list, select **MCID**, in the **MCID Definition** list, select **Inherited from Layup**.

6.7.4. Performing the static analysis

Switch to the simulation model. The model tree of the **Simulation Navigator**

tab shows the created load, constraint, and simulation object containers. The simulation model is ready for solving. Save the model by right-clicking the *Gofra_sim* item in the **Simulation File View** dialog box and choosing **Save**.

To run the solution of the simulation model, right-click *Solution 1* and choose **Solve....** Click **OK**.

Note the many dialog boxes. When the “Completed” message appears in the bottom part of the analysis monitor, and the “Job Finished” message appears in the solution monitor, close all new dialog boxes of the solution.

6.7.5. Viewing static analysis results for the composite product

In the **Simulation Navigator** model tree, double-click the *Results* of active solution *Solution 1* to go to the **Post Processing Navigator** with results loaded. To view the results, expand the *Solution 1* tab by double-clicking *Nodal Displacement*. The nodal distribution of total displacement (absolute value of displacement vector) is displayed in the graphics area.

Use the **Advanced Post Report** command of the **Create Laminate** toolbar. In the *Laminate Post Report*, select *Solution 1*. Click **OK**.

In the **Simulation Navigator** model tree, right-click *Laminate Post Report 1* and choose **Generate Graphical Report**. Select *Solver Ply Stresses and Strains*. Make sure to select the check box for the *Failure Index*. Click **OK**.

Right-click *Graphical Report 1* and choose **Generate Result File** (Figure 6.33).

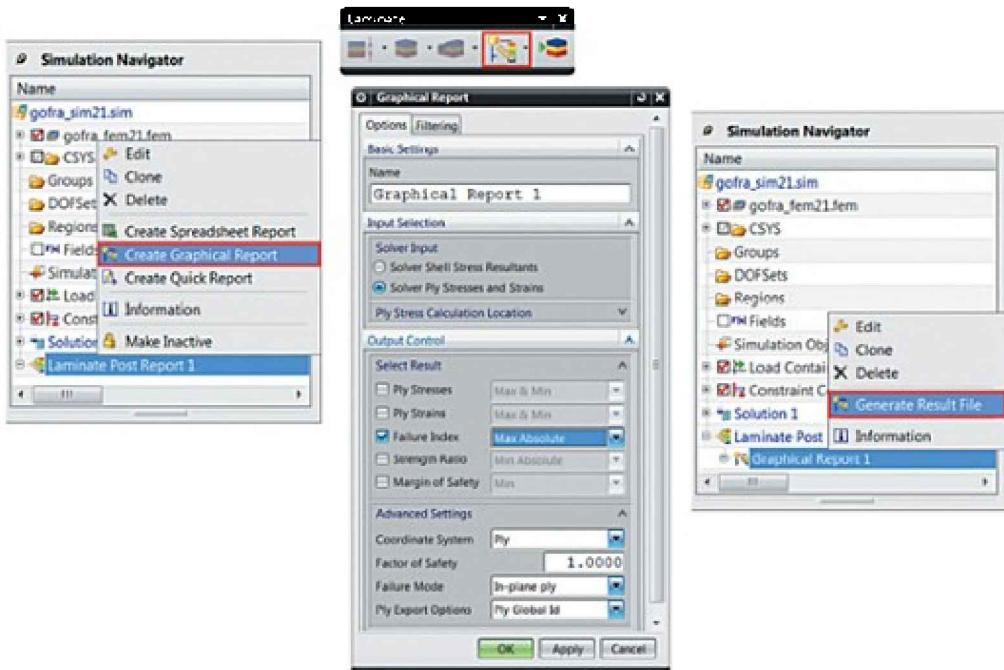


Figure 6.33. Creating an advanced post processing report for composite structures

Double-click the new **Results** node to go to the **Post Processing Navigator** with **Graphical Report 1** results loaded. To view the results, expand the **Graphical Report 1** tab, then double-click **Max Abs FI – Ply 5 p**. The distribution of the failure index is displayed in the graphics area(Figure 6.34).

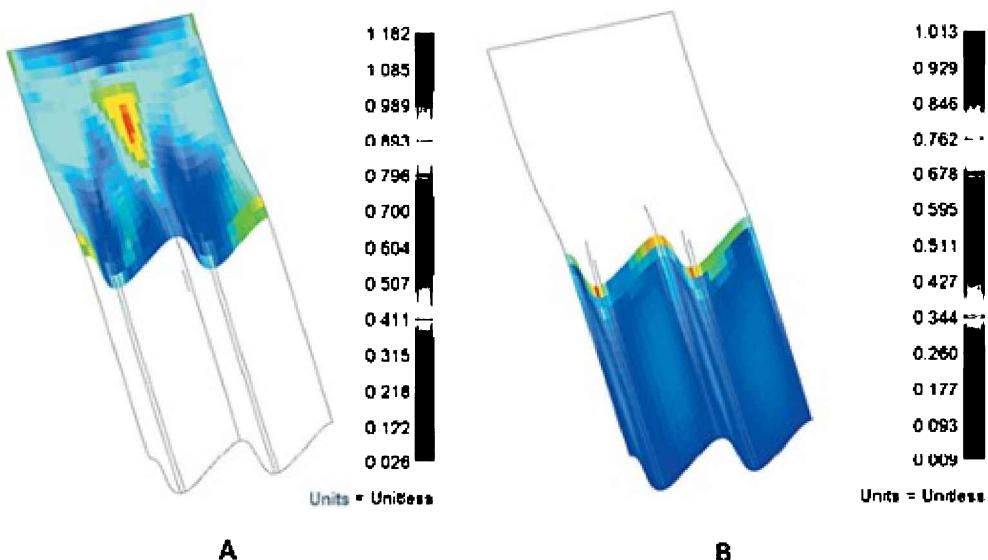


Figure 6.34. Failure index distribution A. For the fifth ply. B. For the third ply.

When you are done working with the model and results, close it by choosing **File→Close→All Parts** in the main menu.

PART

2

Chapter 1. Linear static analysis

Experience shows that most practical problems of deformable solid body mechanics are static analysis problems. Static analysis is used for finding stresses, strains, displacements, and reaction forces in structures under static loads. Static analysis itself is divided into linear and nonlinear analysis because in some situations a structure's behaviour can only be adequately described if its nonlinear properties (nonlinear material behaviour, large strain and displacement, contact interaction, and so on) are taken into account.

Linear static analysis requires less computing resources compared to nonlinear or dynamic problems, so it serves an important auxiliary function. Linear static analysis of structures is useful as a preliminary verification stage before solving nonlinear and dynamic problems. With this approach you can gain a general understanding of the structure's behaviour and control the quality of the FE model.

This chapter describes basic features for linear static analysis of structures implemented in **NX Advanced Simulation** using NX Nastran.

Note that **NX Advanced FEM** pre-processor also supports linear static analysis solutions for third-party solvers:

- *ANSYS - Linear Statics*
- *Abaqus - Static Perturbation*

1.1. Linear static analysis

Linear static analysis of structures involves determining the stress-strain state of statically-loaded structures with specific boundary constraints.

Linear static analysis is performed under the following assumptions:

- The stiffness matrix is not a function of strain.
- Boundary constraints are not functions of strain.
- Any displacements are small.
- Only linear elastic properties of materials are considered.
- Components of the stress tensor and the strain tensor are linearly

related.

- Dynamic effects are neglected.

Linear static analysis involves solving a system of static equilibrium equations for the structure:

$$[K]\{U\} = \{F\}, \quad (1.1)$$

where $[K]$ is the global element stiffness matrix,

$\{U\}$ is the nodal displacement vector,

$\{F\}$ is the external forces vector.

Linear static analysis supports virtually all NX Nastran finite element types except some elements for advanced nonlinear analysis. You can find a more complete list and a summary of NX Nastran finite element types in Chapter 3 of Part 1, and in [25].

You can use isotropic, orthotropic, and anisotropic materials for linear static analysis. The type of material and finite elements is defined by the material properties model with a NX Nastran characteristic card. Available material cards are listed in the following table.

NX Nastran Card	Element types	Material type	Description
MAT1	1D, 2D, 3D, and 2D1/2 (axisymmetric)	Isotropic	Definition of isotropic properties
MAT2	2D	Anisotropic	Definition of anisotropic properties only for shell elements
MAT3	2D, axisymmetric	Orthotropic	Definition of orthotropic properties for axisymmetric elements and for plane stress and plane strain analysis elements
MAT8	2D	Orthotropic	Definition of orthotropic properties only for shell elements
MAT9	3D	Anisotropic	Definition of anisotropic properties for 3D elements
MAT11	3D	Orthotropic	Definition of orthotropic properties for 3D elements

1.2. Methods for solving systems of equilibrium equations

Systems of equilibrium state equations in NX Nastran are solved using direct or iterative methods [11]. The direct methods include:

- Direct right-handed decomposition (Right-Handed Method).
- Direct left-handed Cholesky decomposition (Left-Handed Method).
- Sparse matrix method (Sparse Method).
- Parallel computing method (Parallel Method).

The stiffness matrix is decomposed only once, so for problems with several simulation subcases, direct methods are the most efficient from the point of view of computation and, therefore, time resources. If you need to solve a single simulation case, and the FE model is represented mostly by 3D finite elements, iteration methods are more efficient with regard to performance and RAM usage. These methods can handle very large problems. NX Nastran implements the following iteration methods:

- Global method that uses the full matrix for the entire structure.
- Elemental method that uses element matrices.

You can use the following preliminary equivalent transformations (Preconditioning Methods) for the following methods:

- Block Incomplete Cholesky conjugate gradient decomposition method.
- Jacobi method.
- Cholesky decomposition method.
- Reduced Incomplete Cholesky conjugate gradient decomposition method.

1.3. Solution types for linear static analysis

NX Nastran implements three solution types for linear static analysis:

- *SESTATIC 101 – Single Constraint*. You can use this type to create several simulation cases with the same constraints but different loading.
- *SESTATIC 101 – Multi Constraint*. You can use this type to create several simulation cases with different constraints and different loading for each

individual case.

- *SESTATIC101* – *Superelements*. This solution type can be used to statically reduce mass and stiffness matrices of the general simulation model. This type is used for creating external superelements (substructures) of the structure.

This chapter provides a detailed description of the first two solution types. These types mostly differ in that with *SESTATIC 101 – Multi Constraint*, each simulation case includes its own constraint container.

For example, Figure 1.1 shows the **Simulation Navigator** tab with the simulation model tree that has two solutions: *Single Constraint*, and *Multi Constraint*. Each of these solutions contains two simulation cases.

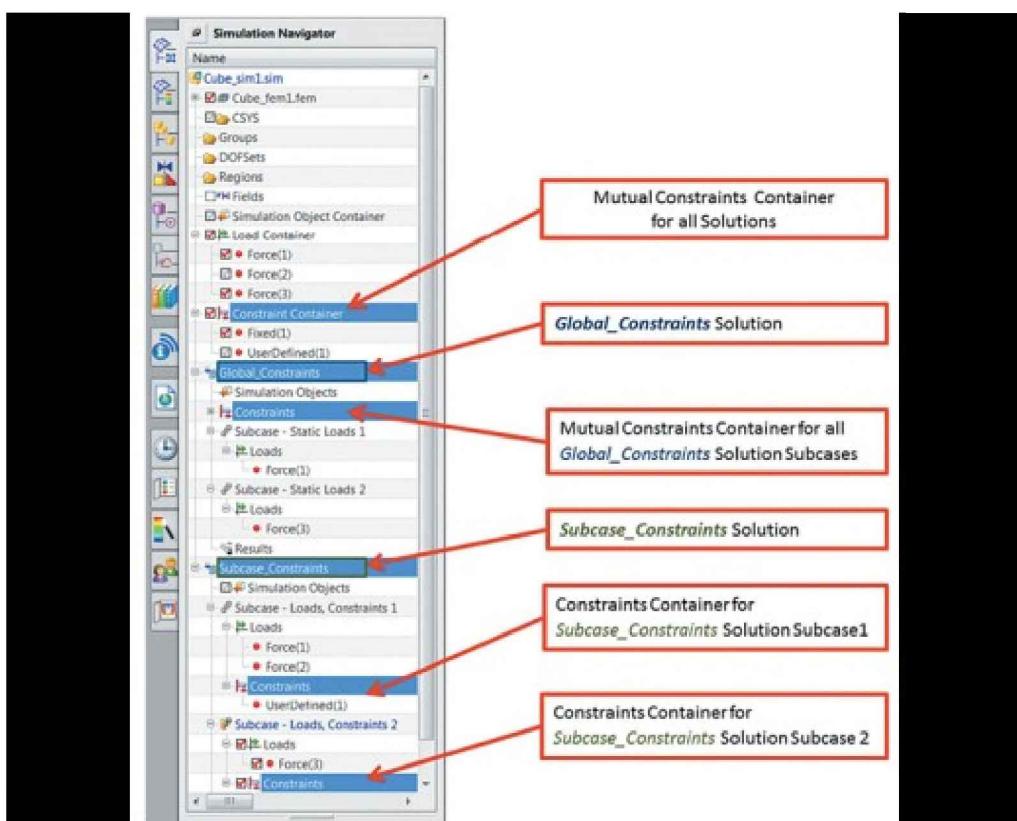


Figure 1.1. Simulation model tree

The simulation model tree is described in greater detail in Chapter 4 of Part 1.

To create a new simulation case, right-click the active solution and choose the **New Subcase** command. The **Solution Step** dialog box opens. It contains simulation case settings that will be described later (Figure 1.2).

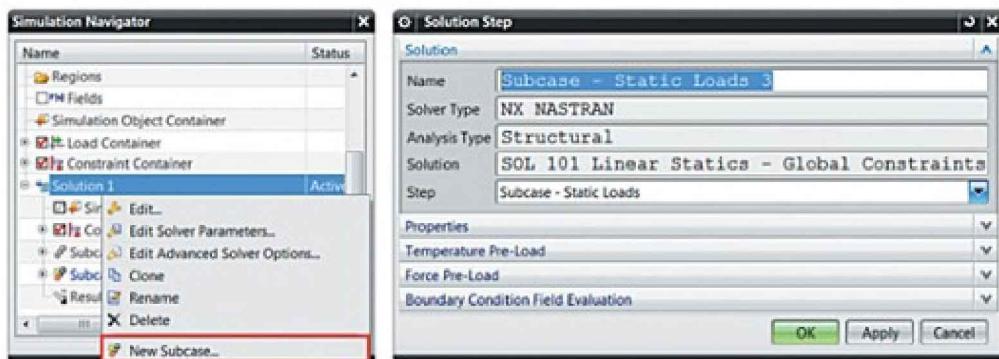


Figure 1.2. Creating a new simulation case

To simplify working with solutions that contain several simulation cases with different constraint sets and loading conditions, **NX Advanced Simulation** implements an additional **Subcase Manager** tool. You can use this tool to control sets of constraints and loads for each simulation case.

To use this tool, right-click the active solution and choose the **Subcase Manager** command in the shortcut menu as shown in Figure 1.3. In the **Subcase Association Manager**, the **Type** list contains, depending on the selected **SESTATIC 101** solution type, the following:

- Only Loads for **SESTATIC 101 – Single Constraint**.
- **Constraints and Loads for SESTATIC 101 – Multi Constraint**.

The **Subcase List** group contains a table with the following columns:

- **Subcase ID**.
- **Subcase Name**.
- Columns with names of all loads or constraints in collectors depending on the selected type.

The number of rows in the table is equal to the number of simulation cases of the active solution. The number of columns is equal to the number of loads or constraints for each simulation case. All loads or constraints used in the simulation case are marked with a green check mark, and the unused ones are correspondingly marked with a red cross mark. To include unused

loads or constraints in the simulation case, simply right-click the red cross mark. To remove loads or constraints, right-click the green check mark.

The columns or rows shortcut menu in **Subcase Manager** additionally provides the following ways of including or excluding loads and constraints in simulation cases of the solution (Figure 1.3):

- You can use the *Set Column On* and *Set Column Off* commands to include or exclude a load or constraint in all simulation cases.
- You can use the *Set Row On* and *Set Row Off* commands to include or exclude all loads or constraints in a single simulation case.
- You can use the *Set All On* and *Set All Off* commands to include or exclude all loads or constraints for all simulation cases of the active solution.

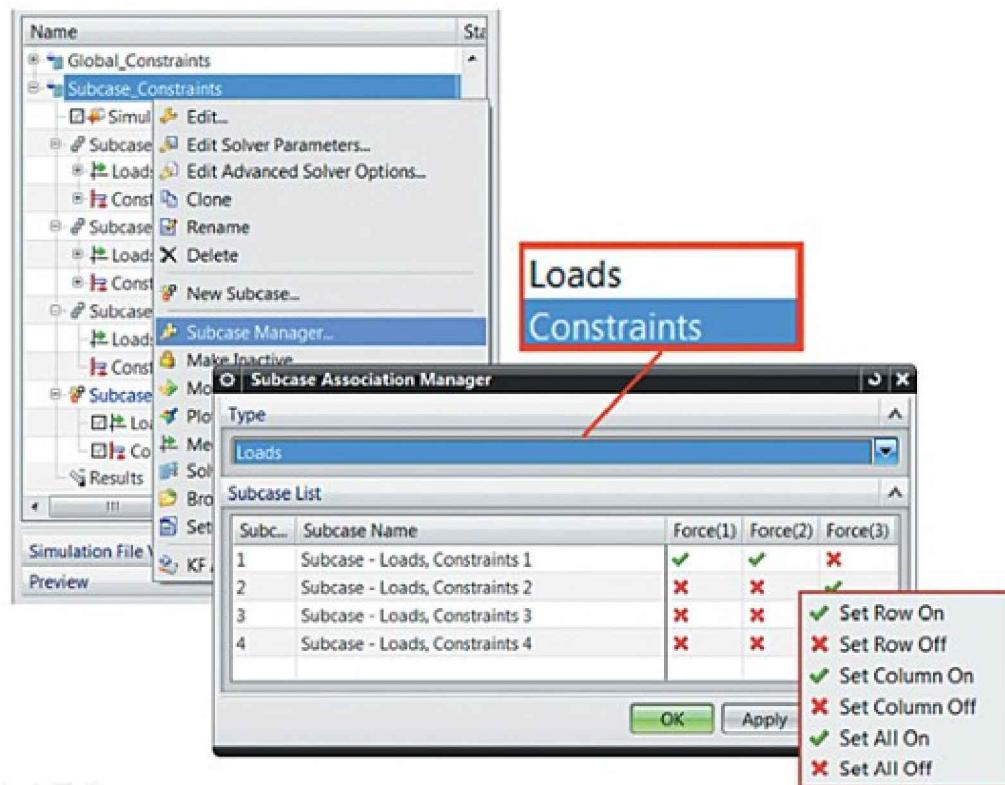


Figure 1.3. Subcase Manager

You can find a detailed description of all types of loads, constraints, and ways to specify them in Chapter 4 of Part 1.

The shortcut menu of the active solution provides the following additional tools (Figure 1.4):

- You can use the *Clone* command to copy the active solution.
- You can use the *Delete* command to delete the solution.
- You can use the *Model Setup Check* command to verify model settings by checking whether physical properties, finite element types, loads and constraints, and simulation objects are set up correctly.
- You can use the *Mechanical Load Summary* command to output total forces. You can select a particular specified load.
- You can use the *Browse* command to directly access the folder where simulation model files are saved.

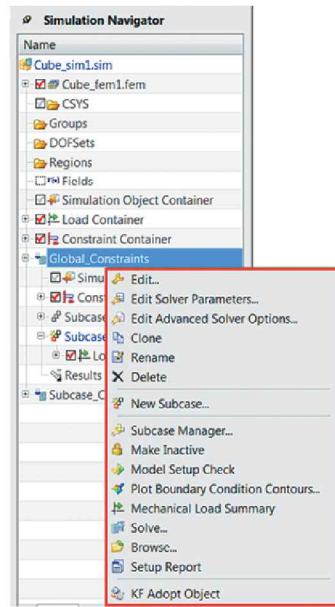


Figure 1.4. Shortcut menu of the active solution

1.4. Setting solution parameters

To create a linear static analysis solution, in the **Simulation Navigator** tab, right-click the name of the simulation model in the simulation model tree and choose **New Solution** in the *shortcut menu*. In the **Solution** dialog box, you can type the name of the new solution, select the *Solver – NX Nastran*, the *Analysis Type – Structural*, and the *Solution Type* – select one of the two solution types for linear static analysis (Figure 1.5).

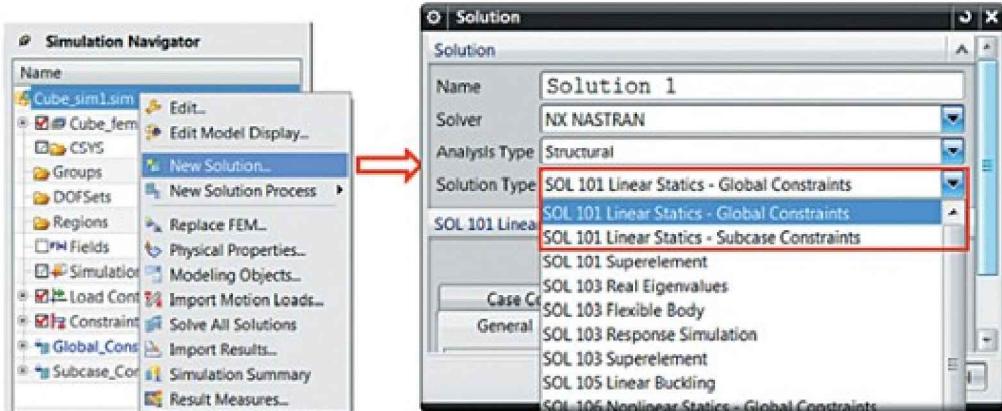


Figure 1.5. Creating a linear static analysis solution

You can also use the **Solution** dialog box to set up solution parameters when you create the new solution or edit an existing solution. Right-click the solution node and choose **Edit**.

All solution parameters you set up in the NX Advanced Simulation interface in the **Solution** dialog box are saved in the *.dat input file of NX Nastran. All tabs in this dialog box except the **General** tab have names that correspond to blocks of the NX Nastran solver input file.

1.4.1. Options of the General tab

The **General** tab includes the following basic parameters and solution options:

1. In the *Description* box, you can enter additional information on the current solution.
2. Under **System Cells**, you can set up NX Nastran *parameters* (Figure 1.6). The default set of system parameter values is suitable for most problems. All system parameters are numbered in accordance with their description [8]. You can add system parameters or edit existing ones using the **Create Modeling Object** command. In the **System Cells n** dialog box, select the relevant parameters and assign desired values, then click **Add**. For example, you can use the **PARALLEL(107)** system parameter to set up parallel computation. Its value reflects the number of CPUs used in the parallel computation.
3. By default, linear static analysis problems are solved applying direct solution methods to systems of equations. To use the iteration method,

select the *Element Iterative Solver* check box. By doing so you make the *Create Modeling Object* command available. Click it to open the ***Element Iterative Solver Options 1*** dialog box and set up parameters of the iteration method.

4. Select the *Ignore Material Temperature Dependence* check box to use the ambient temperature (if set up for the material) or the temperature specified in the *Material Temperature Default* box to determine material properties.
5. Select the *Treat CGAP as Linear Contact Elements* check box to treat all CGAP elements as linear contact elements.
6. The *Boundary Condition Field Evaluation* group is used if you set up loads that vary with time, frequency, or temperature:
 - *Evaluation Time*: use the value for a specific point in time in the calculations.
 - *Evaluation Frequency*: use the value for a specific frequency in the calculations.
 - *Evaluation Temperature*: use the value for a specific temperature in the calculations.
7. Select the *Run Job in Foreground* check box to run the solution in foreground. If this check box is selected, when you run the solution (start NX Nastran), you are unable to work in NX while the computation runs. This also means you cannot monitor the computation process in the *Solution Monitor* window.

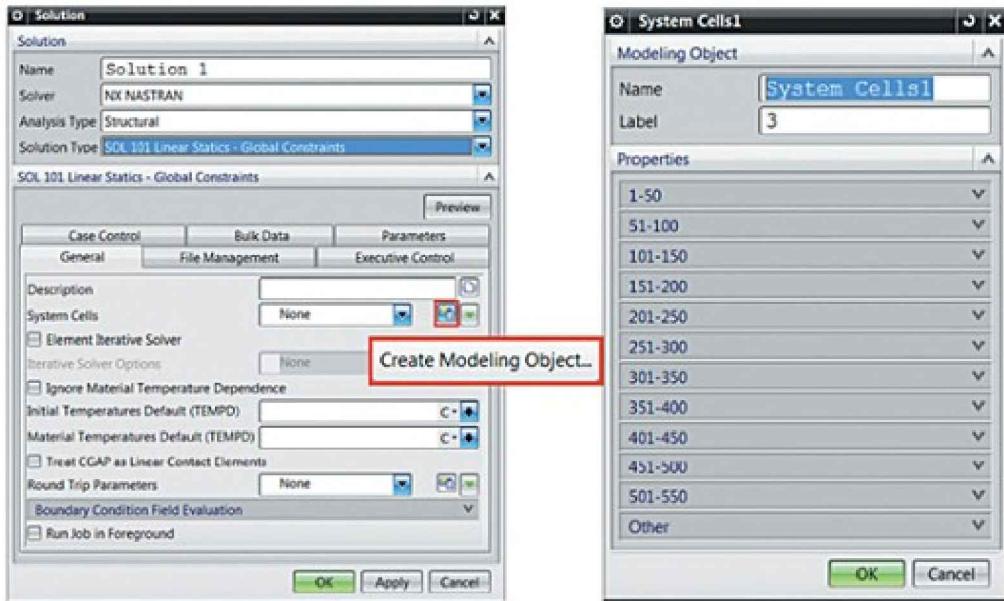


Figure 1.6. Options of the General tab

1.4.2. Options of the File Management tab

The *File Management* tab contains the following options (Figure 1.7):

- In the *User Defined Text* box, you can enter commands that are written into the *FILE MANAGEMENT* block of NX Nastran input file.
- In the *DBset Allocation* box, you can set up options to manage the maximum size and location of the following NX Nastran files:
 - a. *DBALL* – DMAP blocks that are saved for reuse in future runs.
 - b. *SCRATCH* – size and location of temporary data and files.
 - c. *SCR300* – temporary workspace for modules.

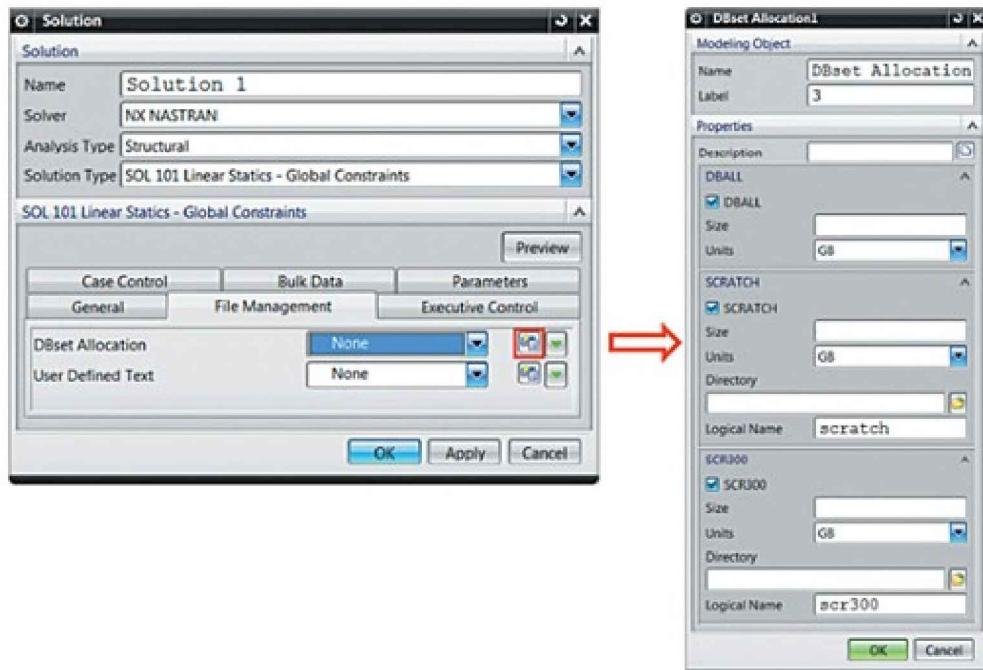


Figure 1.7. Options of the File Management tab

1.4.3. Options of the Executive Control tab

The most important tool you can use for quality control of the FE model in NX Advanced Simulation is the ***Model Check*** command. Its evaluation criteria are associated with threshold values of the solver used. A detailed description of the ***Model Check*** is provided in Chapter 3 of Part 1. Quality control of finite elements is additionally enabled by the ***Geometry Check*** option of the ***Executive Control*** tab in the ***Solution*** dialog box. Default quality criterion values for finite elements are suitable for most FE models in most cases. You can, however, specify your own values. To do so, select ***User Defined*** in the drop-down list and use the ***Create Modeling Object*** command. Then in the ***Geometry Check Options*** dialog box, specify threshold values for various finite element types and set up the message output type: information, warning, or fatal error. If you select ***None*** in the list, the check is not performed.

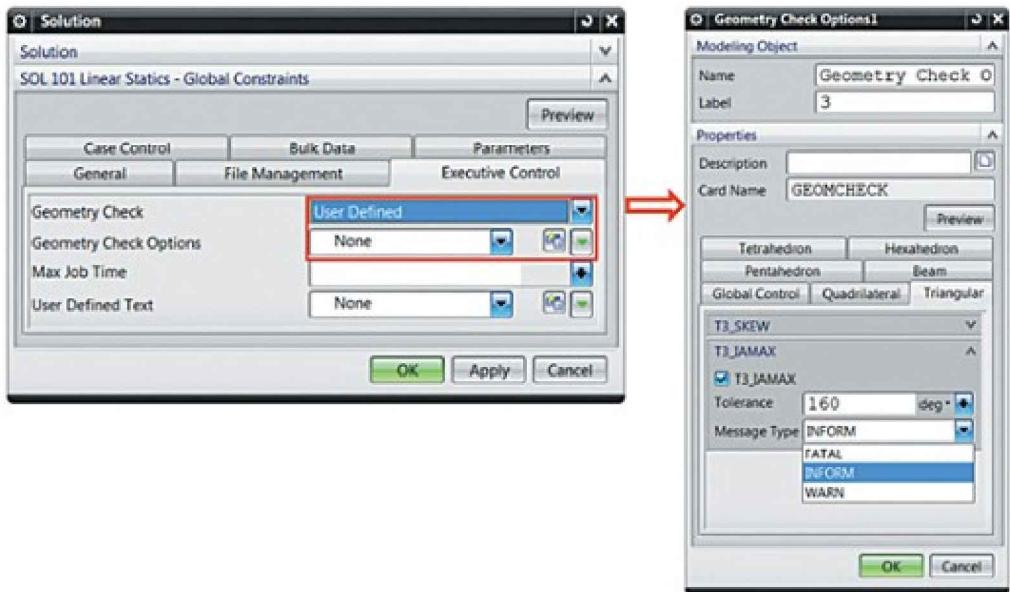


Figure 1.8. Options of the Executive Control tab

To limit solution time, you can enter a time value in seconds in the *Max Job Time* box. The computation is stopped when this time elapses, regardless of results.

You can write additional commands into the *EXECUTIVE CONTROL* block of the NX Nastran input file using the *User Defined Text* box.

1.4.4. Options of the Case Control tab

The *Case Control* tab of the **Solution** dialog box is used to set solution parameters in the *CASE CONTROL* block of the NX Nastran input file. This tab includes the following options (Figure 1.9):

1. You can use the *Bulk Data Echo Request* option to control the output of the *BULK DATA* block of the input file into the *.f06 results file.
2. You can use the *Rigid Body Checks* option to check the model for possibility of movement similar in character to rigid body movement [4, 8].
3. You can use the *Rigid Element Method* option to select the rigid element (*RBAR*, *RBE1*, *RBE2*, *RBE3*, and *RROD*) processing method:
 - Select *Linear Elimination* to disregard thermal expansion. This method is used by default.
 - Select *Lagrange Multiplier* to take thermal expansion of rigid elements into account.

4. You can use the *Output Requests* option to set up the set of values for writing into the *.op2 results file and subsequent processing in the **Post Processing Navigator** tab. By default in NX Advanced Simulation, the **Structural Output Request1** modeling object is set up. This modeling object includes the following set of values:
 - *Displacement*.
 - *Stress*.
 - *SPC Forces*, that is, reaction forces in constraints.

You can create a new set of values for output into the results file with the **Create Modeling Object** command. To edit an existing set, you can use the **Edit** command. When you use these commands, the **Structural Output Request n** dialog box opens. This dialog box contains a set of tabs with results output settings, one tab per value (Figure 1.9). To add a value to the results set, select the tab with the same name then select the relevant check box, for example for displacement, *Enable Displacement Request*. Then you can choose the method of writing information into the *.f06 results file, the data output format, and the sorting type. You can also select a limited group of finite elements for which the result is output.

The **Structural Output Request n** dialog box provides the following additional commands:

- You can use the **Preview** option to open the *Information* dialog box with a list of all values selected for output.
 - You can use the **Enable All** option to select all values.
 - You can use the **Disable All** option to cancel the selection of all values.
5. The *Global Contact Parameters* are discussed in the context of linear contact in para. 1.5.2.
 6. The *Global Glue Parameters* are discussed in para. 1.6.1.
 7. You can write additional commands into the CASE CONTROL block of the NX Nastran input file using the *User Defined Text* box.

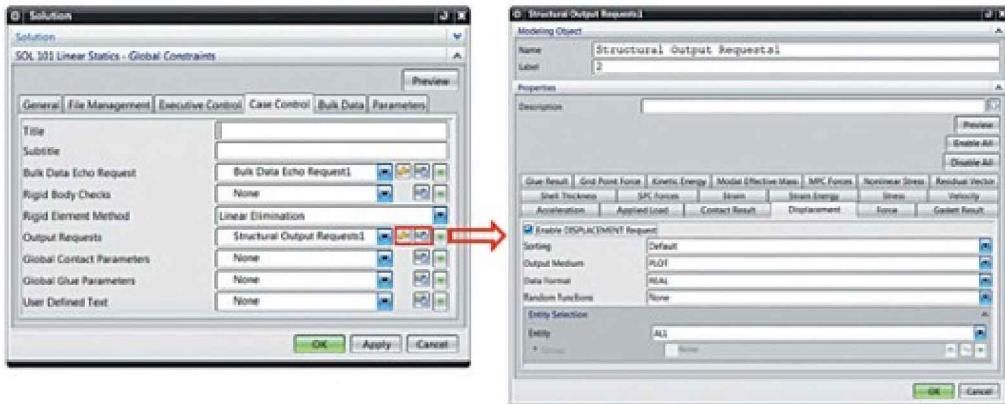


Figure 1.9. Options of the Case Control tab

1.4.5. Options of the Database tab

You can use the *Print CWELD/CFAST Diagnostic Information* if the model contains CFAST and CWELD connections (modeling point welds and rivet joints). If you select this parameter, additional information is written into the *.f06 file whenever errors occur in these connections.



Figure 1.10. Options of the Database tab

You can use the *DOF Sets* group to create sets of degrees of freedom.

This feature is typically used in dynamic analysis problems, when you need to create super elements and use them in FE assemblies [8].

You can write additional commands into the *Bulk Data* block of the NX Nastran input file using the *User Defined Text* box.

1.4.6. Specifying additional solution parameters

You can select and set up additional NX Nastran parameters in the **Solution Parameters n** dialog box. To open this dialog box, use the **Create Modeling Object** command in the **Parameters** tab. Parameters in this dialog box are grouped alphabetically by initial letter of parameter name (Figure 1.11). In the **Solution Parameters n** dialog box, select parameters of interest, assign values as necessary, then click **Add**. After you add the parameters, the **Solution Parameters n** dialog box displays an additional **Specified Properties** group with user-defined parameters.

A complete list of parameters with descriptions is provided in [8].

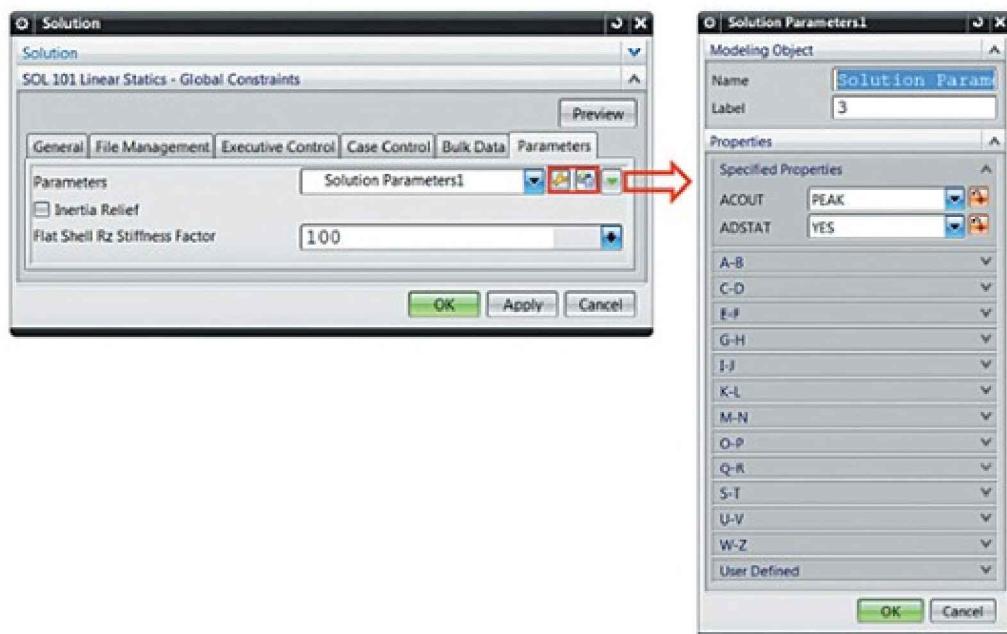


Figure 1.11. Specifying additional solution parameters

Solving static analysis problems using the FE method assumes impossibility of solid motion of the model without strain. Otherwise, the stiffness matrix becomes singular, and the solution is aborted or physically meaningless results are obtained.

However, NX Nastran can simulate unconstrained structures. This is necessary, for example, to study the behavior of spacecraft in orbit, aircraft in flight, and so on. In such cases inertial balancing is used. This approach is based on using the mass of the model to create inertial forces that balance the applied loads, bringing the model to equilibrium. To use this approach,

select the *Inertia Relief* check box in the *Parameters* tab of the **Solution** dialog box.

The *Flat Shell Rz Stiffness Factor* parameter has a default value of 100. This parameter prevents infinite nodal values of the rotational degree of freedom when 2D shell elements are used. The value of this parameter is added to the rotational stiffness along the normal and is only applicable to CQUAD4, and CTRIA3 element types.

You can perform final monitoring of all specified commands, parameters, and options of the solution using the **Preview** command of the **Solution** dialog box. As a result, the *Information window opens*, containing a segment of the NX Nastran input file.

1.4.7. Specifying solution parameters for the simulation case

You can specify solution parameters for each individual subcase. To do so, choose *Edit in the shortcut menu of the simulation subcase*. In the **Solution Step** dialog box, you can set up the following options (Figure 1.12):

- Under *Output Request*, you can change the set of values that is output in the results of the specific simulation subcase.
- In the *Case Control User Defined Test* box, you can specify additional commands for the *Case Control* block of the NX Nastran input file.
- In the *Temperature Pre-Load* group, you can use thermal simulation results obtained from other solvers (NX Thermal, Abaqus, Ansys) as the pre-load.
- In the *Force Pre-Load* group, you can use pressure fields obtained from gas dynamics simulations in NX Flow as the pre-load.

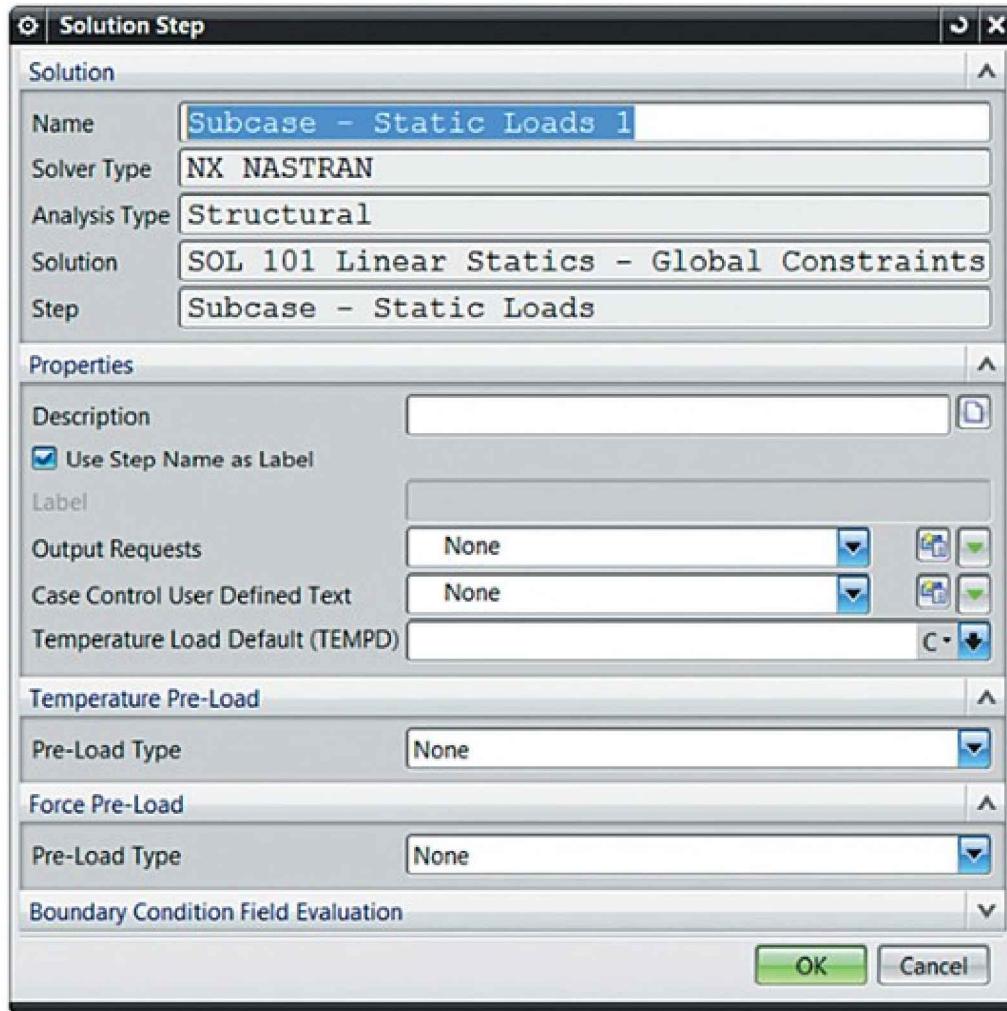


Figure 1.12. Specifying solution parameters for the simulation case

1.5. Linear contact interaction

You can use linear static analysis solutions to model linear contact interaction of surfaces.

All created contact interactions are simulation objects of the **Surface-to-Surface Contact** type. These objects are set up using the **Advanced Simulation** toolbar in the **Simulation Object Type** command list. The procedure of creating simulation objects such as **Surface-to-Surface Contact** is described in detail in para. 4.3.3 of Part 1. Note that contact surfaces can be specified automatically or manually within the given search range. The contact surfaces are subdivided into **Source Regions** and **Target**

Regions.

Contact interaction conditions for free faces of finite elements on the touching surfaces are automatically created by the solver when you run the solution of the simulation model if the following conditions are satisfied:

- Free faces of finite elements on the contacting surfaces intersect.
- The length of the normal segment between the faces of the elements is less than or equal to the value of the search range.

Unlike non-linear contact where contacting element pairs are redefined at each iteration, in linear static problems, pairs are created once in the beginning of the simulation and are not updated afterwards.

1.5.1. Linear contact fundamentals

The following is a summary of linear contact interaction in NX Nastran [4].

The following relations are used by the contact algorithm:

- Kinematic equations that describe the movement of surfaces relative to each other.
- Linear contact conditions.
- Static equilibrium conditions with added contact stiffness and contact forces.
- Restatement of kinematic equations and fundamental equations (equilibrium and contact conditions) in terms of FE.

Figure 1.13 shows free faces of elements of the target (A) and source (B) surfaces. As the model is deformed, the point on the source face (acting point, D) moves relative to the target face. The solution involves constructing three mutually perpendicular unit vectors through the acting point D: tangential vectors e_1 and e_2 , and vector n that is normal to the source face. The point where the normal vector n intersects the target face is the target point C. The contact algorithm assumes that the acting point penetrates the target face at target point C. The penetration value is then calculated as follows:

$$p = p_0 + (u_h - u_i)n, \quad (1.2)$$

where p_0 is the initial penetration determined by the original undeformed geometry,
 u_h is the displacement (movement) of the acting point D,
 u_t is the displacement (movement) of target point C.

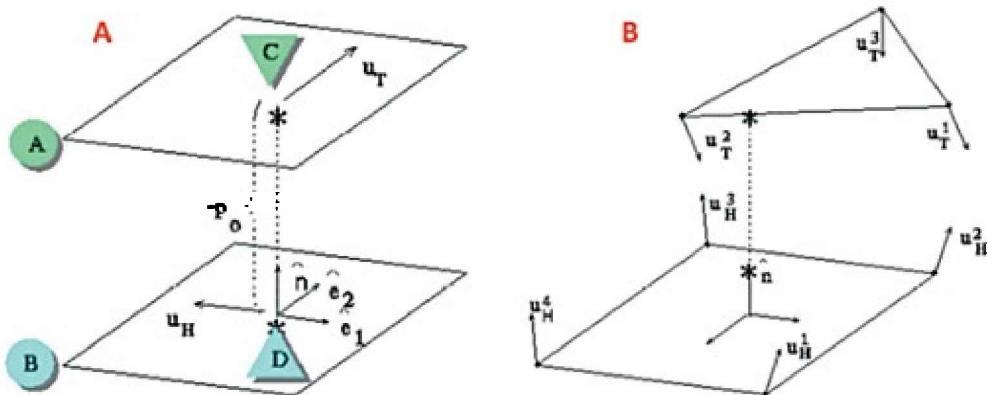


Figure 1.13. Interaction diagram for the contact pair of elements

To account for Coulomb friction, it is necessary to determine the relative increment of tangential displacement:

$$\Delta u_t = (\Delta u_h - \Delta u_v) - [n(\Delta u_h - \Delta u_v)] n. \quad (1.3)$$

In terms of NX Nastran, the contact pair is the totality of interaction of the acting and target points, as well as the nodal degrees of freedom of source and target faces of elements (Figure 1.13, B).

Penetration of a particular contact pair is then defined through interpolation functions for nodes of the source and target element faces and is of the following form:

$$p = p_0 + [q_n] \begin{pmatrix} u_H \\ u_T \end{pmatrix} \quad (1.4)$$

where $[q_n]$ is a row matrix consisting of a product of interpolation functions and normals.

u_H and u_T are nodal displacements of the source and target faces.

The analogous equation for tangential sliding is of the following form:

$$\Delta \xi = \xi_0 + [q_t] \begin{pmatrix} u_H \\ u_T \end{pmatrix} \quad (1.5)$$

For all contact elements, expressions (1.4) and (1.5) take the following form in matrix notation:

$$\{p\} = [Q_n]\{U\} + \{P_0\} \text{ and } \{\Delta \xi\} = [Q_t]\{U\} + \{\xi_0\}. \quad (1.6)$$

Solving the contact interaction algorithm involves satisfying the following linear contact condition:

$$t_n p = 0, \quad (1.7)$$

where $p \leq 0$ is the zero interpenetration condition of the contact surfaces.

$t_n \geq 0$ is the contact pressure (transmission of contact forces between contacting bodies).

Condition (1.6) is satisfied in two cases:

1. $p = 0$ with $t_n \geq 0$ – zero interpenetration.
2. $t_n = 0$ with $p \leq 0$ – the bodies do not touch.

If Coulomb friction must be considered, condition (1.8) also needs to be satisfied:

$$|t_t| - \mu t_n \leq 0, \quad (1.8)$$

where t_t is the frictional force,

μ is the Coulomb friction coefficient,

if $|t_t| < \mu t_n$, contacting surfaces are locked.

If $|t_t| = \mu t_n$, the contacting surfaces “slide” along each other.

In order to solve linear contact interaction, NX Nastran uses the augmented Lagrange method [4]. Unlike the penalty function method, in addition to introducing penalty stiffness values into the global stiffness matrix, the linear contact condition (zero interpenetration) is satisfied by adjusting contact forces t at each iteration of the linear contact algorithm using the following expression (1.9):

$$t^i = t^{i-1} + \varepsilon p, \quad (1.9)$$

where t is the contact force,

p is the current penetration value at the point,

ϵ is the penalty stiffness.

The solution of the linear contact algorithm comprises two iterative loops: “outer” and “inner” [4]. The “outer” loop determines the status of contact elements. The “inner” loop iteratively updates contact forces to satisfy the zero penetration condition for all active contact elements.

The following is the linear contact interaction solution algorithm:

1. Creation of contact elements based on user-specified areas and the given search range.
2. Calculation of element stiffness characteristics, including penalty stiffness values of contact elements.
3. Determination of the signs of interaction forces for elements taking into account boundary conditions and applied loads.
4. Entering the “outer” loop to determine contact element status depending on interpenetration and the contact force.
5. Adding contact stiffness to the global stiffness matrix of active contact elements.
6. Entering the “inner” loop. Each iteration of the “inner” loop determines penetration in contact pairs. This value is used in expression (1.9) to calculate contact forces for the next iteration in accordance with the specified normal (*PENN*) and tangential (*PENT*) values of the penalty function. The “inner” loop is completed when either the maximum number of iteration (*MAXF*) is reached or the contact force convergence condition is satisfied, that is, the change of contact forces between the previous and next iterations is less than or equal to the *CTOL* parameter. After completion, the flow returns to the “outer” loop.
7. In the “outer” loop of the algorithm, the contact forces calculated in the “inner” loop are used to update the status (activate or inactive) of contact elements. If any penetration of inactive contact elements is found, their status changes to active. Thus a new set of active contact elements is created, and the “inner” loop of the algorithm is executed again.

The solution of the linear contact algorithm is considered complete and converged when the following conditions are satisfied:

- The convergence condition for the *CTOL* parameter is satisfied, and the number of contact element status change events is less than or equal to the specified threshold value of the *NCHG* parameter.
- The number of iterations of the “outer” loop reaches the maximum number of iterations (*MAXS*) as specified in the “outer” loop settings.

1.5.2. Linear contact parameters

NX Nastran uses the following parameters for the linear contact interaction solution algorithm. You can set global linear contact parameters that are used for all simulation objects of the *Surface-to-Surface Contact* type, as well as local parameters assigned to specific simulation objects. The values of local parameters have precedence for a particular simulation object.

You can specify local parameters when you create simulation objects of the ***Surface-to-Surface Contact*** type in the dialog box of the same name (Figure 1.14). If you need to model contact interaction with friction forces, set up the *Coefficient of Static Friction* parameter. By default, this coefficient is equal to zero, that is, any friction forces are neglected. In the *Local Contact Pair Parameters* group, you can specify parameters for linear and nonlinear contact. This chapter discusses parameters for linear contact: *Linear Overrides* (*BCTPARM*). To specify a local parameter set, use the *Create Modeling Object* command. In the ***Contact Parameters – Linear Pairs overrides n*** dialog box, you can find the following parameters:

- In the *Penalty Normal Direction* (*PENN*) box, you can specify the normal component of the penalty function.
 - In the *Penalty Tangential Direction* (*PENT*) box, you can specify the tangential component of the penalty function.
 - In the *Penalty Factor Units* (*PENTYP*) box, you can enter the unit of measurement for penalty functions:
- a. Select *1/Length* (default) to calculate the contact stiffness of the element using the following expression:
- $$K_{\text{cont}} = e * E * dA, \quad (1.10)$$
- where *e* is the *PENN* or *PENT* parameter,

E is the elastic modulus of the material of the contacting bodies,
 dA is the area.

The physical equivalent of contact stiffness is the axial stiffness of a rod that has cross section area dA , elastic modulus E , and length $1/e$.

- b. Select $Force/(Length \times Area)$ – equivalent to the spring stiffness to area ratio. In this case contact stiffness of the element is calculated using the following expression:

$$K_{cont} = e * dA. \quad (1.11)$$

- In the *Initial Penetration/gap (INIPENE)* box, you can set up the initial state of the contact surfaces. This parameter is useful when the contact surfaces are curvilinear, and after finite-element approximation of geometry, the results do not lie exactly on the surface (Figure 1.15, A). This in turn can lead to locally increased contact pressure instead of uniform smooth distribution. The following options are possible:
 - a. *Calculate from geometry* is the default option: all gaps and penetrations are accounted for using coordinates of finite element nodes (Figure 1.15, B). This option is convenient for modeling contact interaction with pretensioning where penetration of one body with the other is intentionally defined.
 - b. *Ignore penetration/Use gaps*: any penetration is treated as touching, that is, no additional stiffness penalty functions are introduced for these areas (Figure 1.15, C).
 - c. *Set to zero*: within the search range, all gaps and penetrations are treated as touching.

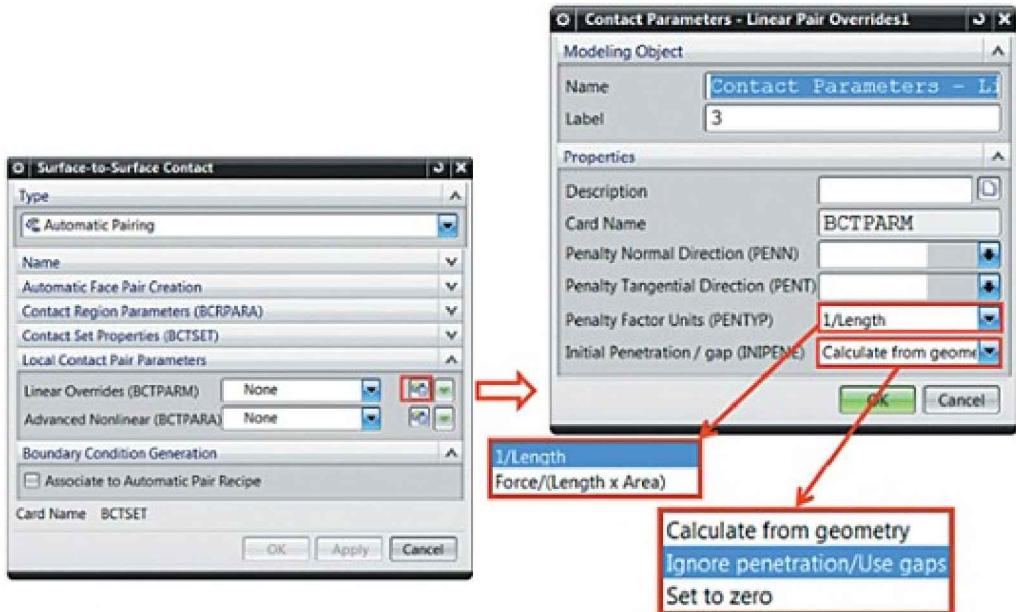


Figure 1.14. Specifying local parameters of the contact pair

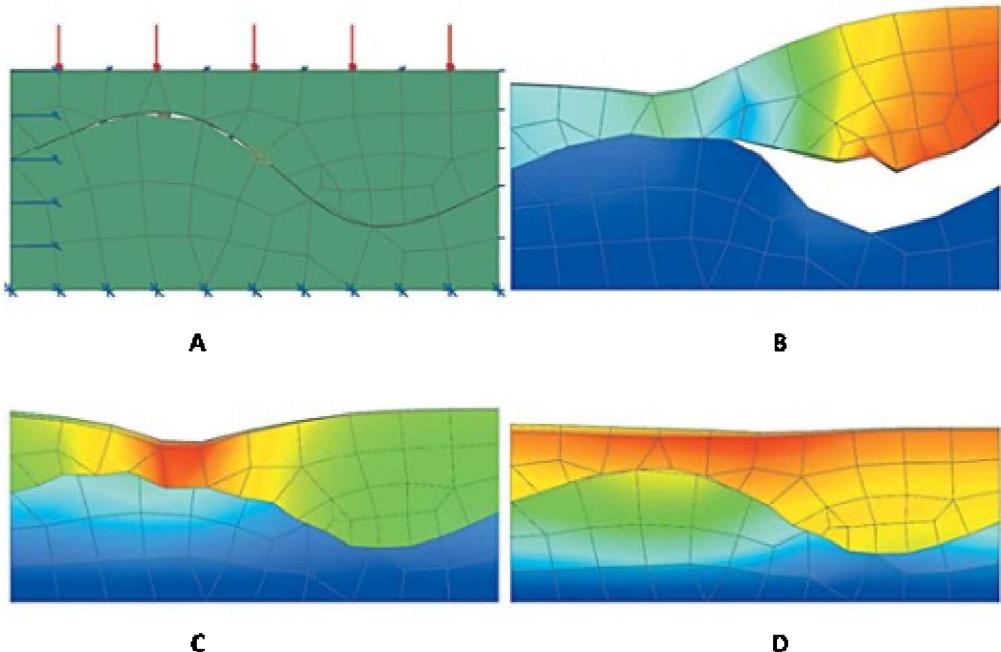


Figure 1.15. Initial penetration and gaps options

NX Nastran additionally supports global parameters of the linear contact algorithm. You can specify the global parameters in the Case Control tab of the **Solution** dialog box. To specify global parameters of linear contact, use

the **Create Modeling Object** command for *Global Contact Parameters*. to open the **Contact Parameters – Linear Global n** dialog box (Figure 1.16). The *Penalty Normal Direction (PENN)*, *Penalty Tangential Direction (PENT)*, *Penalty Factor Units (PENTYP)*, *Initial Penetration/gap (INIPENE)* parameters are analogous to local parameters of a Surface-to-Surface Contact simulation object. Additional parameters:

- The *Contact Force Tolerance(CTOL)* criterion of contact force convergence as determined from the difference of the contact forces between the previous and the next iterations of the “inner” loop of the linear contact algorithm. The convergence of the “inner” loop is considered achieved when the difference of the contact forces is less than the *CTOL* value.
- *Max Iteration Force Loop (MAXF)* is the maximum allowable number of iterations of the “inner” loop of the contact algorithm.
- The *Contact Changes for Convergence* option defines the format of *NCHG* convergence criterion for the “outer” loop. Depending on the selected format, the name of *NCHG* criterion changes:
 - a. *Percentage of Active Contacts – Percentage of Active Contact Elements (NCHG)*: you enter a value less than one, that is, a fraction of the number of active contact elements. In this case the value of the allowable number of contact status change events is different at each iteration of the “outer” cycle because it is calculated using the number of contact pairs with an active status.
 - b. *Number of Contact Changes – Allowable Number of Contact Changes (NCHG)*: you enter an integer value, that is, you specify the allowable number of contact element status changes. In this case the value of the criterion of allowable contact status change events is not modified from iteration to iteration of the “outer” loop, and is equal to the value entered in this box.
- The *Shell Thickness Offset (SHLTHK)* parameter is used for FE modeling with 2D shell elements. If you specify *Include*, the thickness of 2D shell elements is taken into account, that is, contact is achieved at half-thickness distance. If you specify *Exclude*, the thickness is neglected.
- The *Contact Status (RESET)* option is used if the solution contains

several simulation cases. To save computing time, on the first iteration of the “outer” loop, you can use results of the contact pairs obtained when solving the first simulation case. To do so, select the *Start from Previous option*. If you select *Start from Initial*, the status of contact elements is determined anew for each simulation case.

- The *Refine Source Region Mesh (REFINE)* option rebuilds the finite-element mesh of the contact surface as the algorithm is executed to create the most suitable splitting.
- The *Number of Contact Evaluation Points (INTORD)* option defines the number of points on a single face of the original element of the surface for contact calculation purposes. You can obtain more precise results by increasing the number of calculation points but the computation time increases correspondingly. This number of points depends on the *INTORD* parameter and the type of the element face.

	Number of points		
Element face	INTORD=1	INTORD=2	INTORD=3
Linear Triangle	1	3	7
Parabolic Triangle	3	7	12
Linear Quad	1	4	9
Parabolic Quad	4	9	16

- The *Include Shell Element Z- Offset (ZOFFSET)* option takes into account offset parameters specified for 2D shell elements.
- The *All Contact Can Become Inactive (CSTRAT)* option reduces the likelihood of singularity due to inactivation of all contact pairs when you select *Likelihood reduced by the software*.

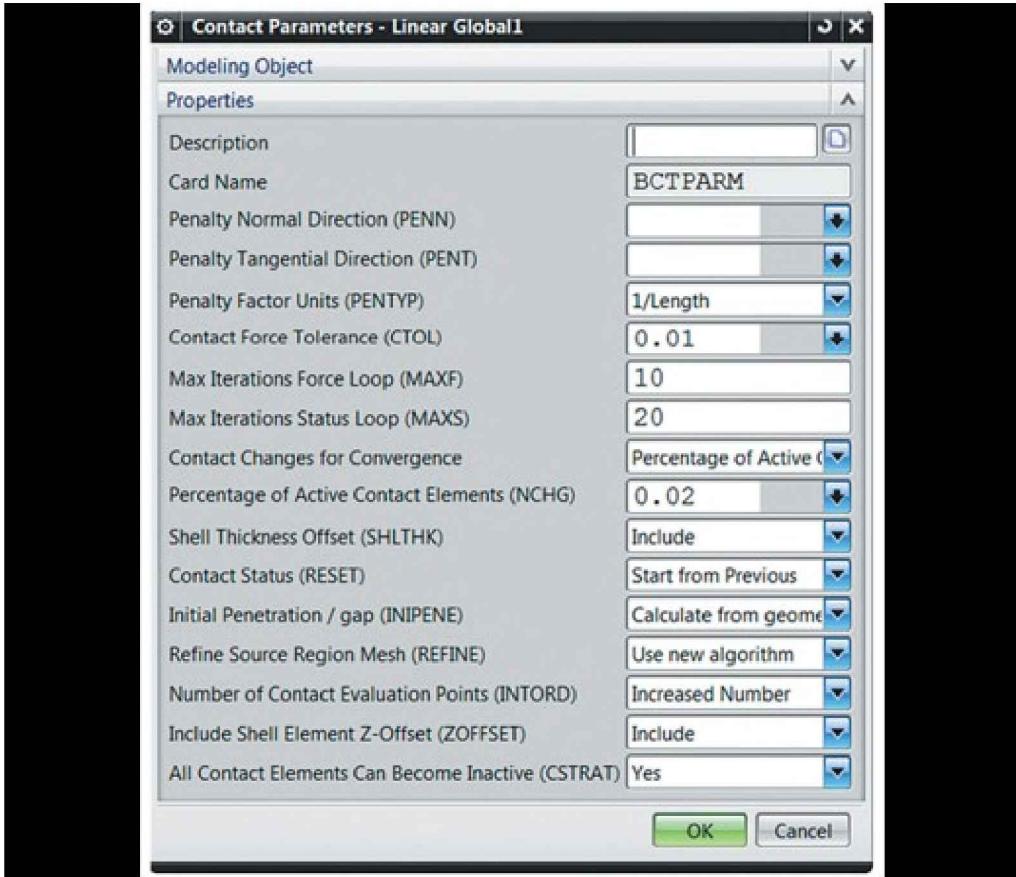
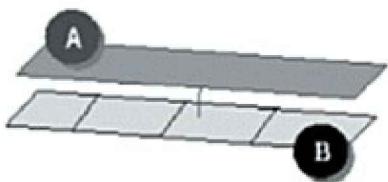


Figure 1.16. Setting up global linear contact parameters

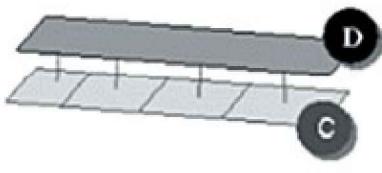
Experience shows that default values of linear contact parameters are suitable for most problems. Linear contact parameters are described in greater detail in [4, 8].

1.5.3. Recommendations for linear contact solutions

When you generate a contact element (a contact pair of element faces), a normal is extended from each free face of the source surface finite element towards the face of the target surface element. Therefore, when there are several elements of the target surface per element of the source surface (Figure 1.17, A), only one contact element is created. When several elements of the source surface correspond to a single element of the target surface (Figure 1.17, B), several contact elements are created. Thus, to accurately describe contact interaction, the characteristic size of the source surface FE mesh should preferably be smaller than the size for the target surface.



A



B

Figure 1.17. Creating contact pairs with different FE sizes on contacting surfaces

When you specify contact regions on a complex (irregular) geometry, you should specify several contact pairs (Figure 1.18).

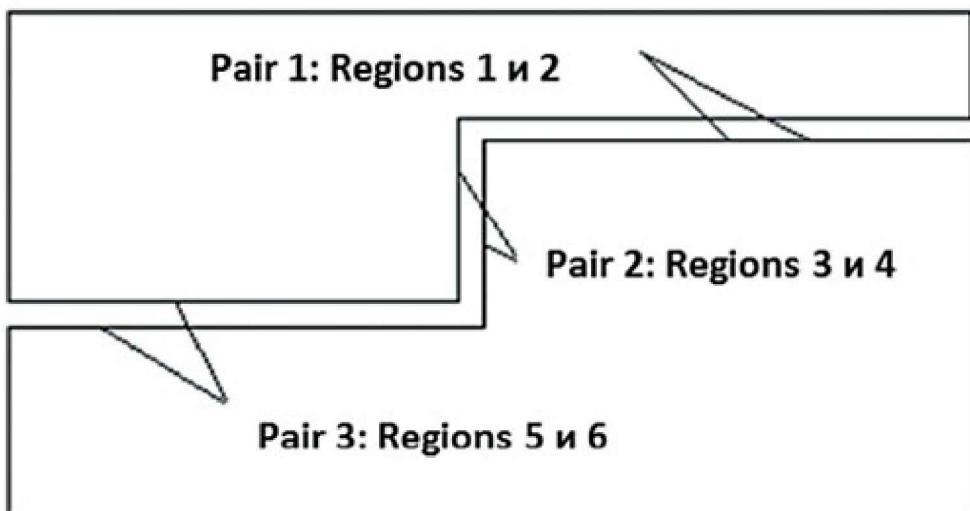


Figure 1.18. Creating several contact pairs on complex geometry

When selecting contacting regions, you should avoid using the same element face for more than one contact zone. Figure 1.19, A shows an example where contact zones 2 and 3 have a common element face. In some situations this can lead to increased computation time. In this case the contact zone arrangement shown in Figure 1.19, B is preferable.

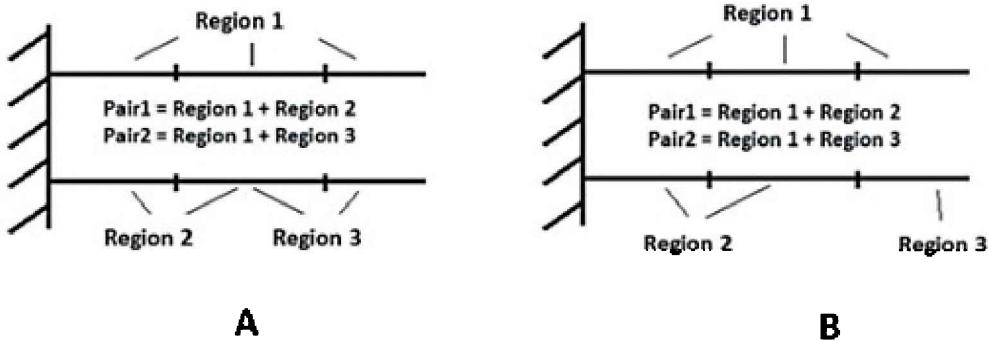


Figure 1.19. Specifying the zone for a complex contact pair

1.5.4. Linear contact results output

In addition to values specific to linear static analysis, the solution results of a problem with linear contact include the following values (Figure 1.20):

- *Contact Traction - Nodal* is the friction-traction force caused by friction or the special properties of contact surfaces.
- *Contact Pressure - Nodal* is the contact pressure at nodes.
- *Contact Force - Nodal* is the nodal forces value.
- *Initial Contact Separation – Nodal* and *Final Contact Separation– Nodal* correspond to the distance between elements in active contact pairs in the beginning and the end of the solution.

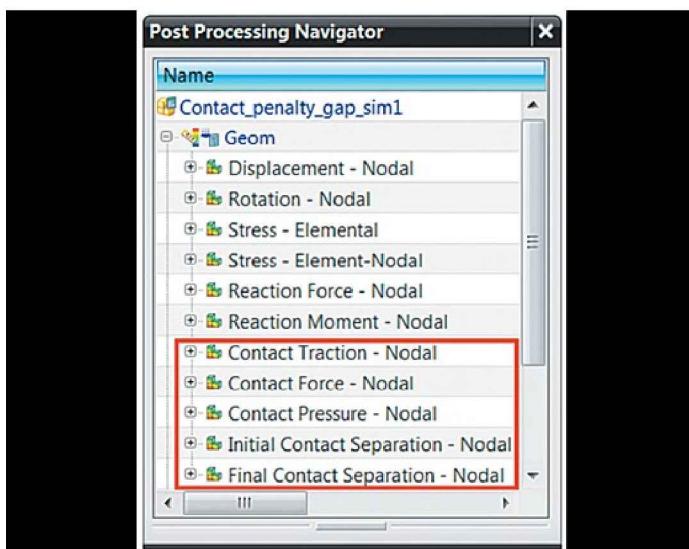


Figure 1.20. Linear contact results in the Post Processing Navigator tab

To output these results, in the **Solution** dialog box, in the **Case Control** tab, use the **Output Requests** list. Either change the *Modeling Object currently used*, or create a new one. In the **Structural Output Requests n** dialog box, in the **Contact Result** tab, select the *Enable BCRESULTS Request* check box. Then select values to output into the *.op2 results file for viewing in the postprocessor (Figure 1.21).

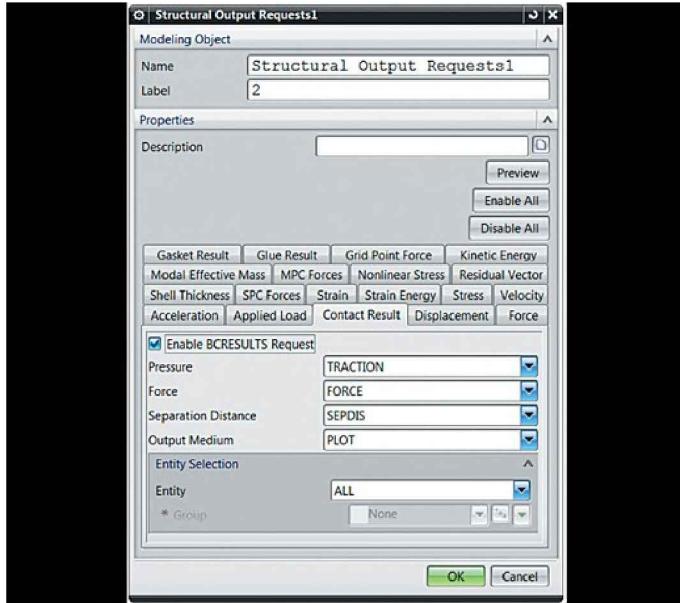


Figure 1.21. Setting up linear contact results output into the *.op2 results file

1.6. Gluing connections

Deformable solid body mechanics problems often require limiting relative displacement of modeling objects in all directions. For example, if elements of the structure are welded or bolted together, however, detailed FE modeling of the connection itself is not necessary, and only transmission of loads from one element to the other is relevant. The connected elements of the structure also have mismatching FE meshes on the interaction boundaries. To model connections of this kind in NX Advanced Simulation, you can use **Surface-to-Surface Gluing** and **Edge-to-Surface Gluing** simulation objects. The creation procedure for simulation objects of this type is described in detail in Chapter 4 of Part 1. You can set up regions of connection (hereinafter referred to as gluing) automatically or manually within the specified search range. A gluing region can be either a *Source Region* or a *Target Region*.

Gluing is achieved by creating stiff springs or connections equivalent to point welds between free faces of elements. In terms of NX Nastran, the combination of free faces of finite elements in the source and target gluing regions together with stiff springs created by the solver is called a gluing element. A pair of free faces of finite elements in the source and target gluing regions is called a gluing pair. Gluing is generated by adding large normal and tangential penalty stiffness values to the gluing elements.

Gluing elements are automatically created by the solver if the following conditions are satisfied when the simulation model is solved:

- The free faces of finite elements in gluing regions intersect.
- The length of the normal between the faces of the elements is less than or equal to the value of the search range.

If you use the ***Surface-to-Surface Gluing*** simulation object, you need to specify the target and source surfaces. Unlike the linear contact case, the designation of the source and target faces is inconsequential.

If you use the ***Edge-to-Surface Gluing*** simulation object, you need to specify free faces of 3D elements or shell elements as the target surface, and edges of shell elements as the source region. In this case the solver automatically creates pseudosurfaces along the specified edges of the source region. The solver then creates a weld-type connection between these pseudosurfaces and the target surfaces.

The example in Figure 1.22 shows a target surface (green line) glued to an edge of the source region (blue line). The created pseudosurface is shown in red, and t is the thickness of shell elements of the source region.

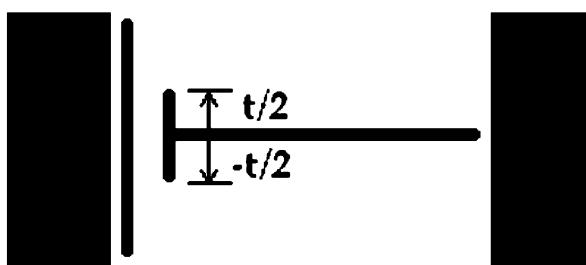


Figure 1.22. Edge-to-surface gluing schematic

1.6.1. Gluing parameters

In NX Advanced Simulation, you can specify global gluing parameters that are used for all simulation objects of the **Surface-to-Surface Gluing** and **Edge-to-Surface Gluing** types, as well as local parameters assigned for each particular simulation object. The values of local parameters have precedence for a particular simulation object.

To specify local parameters, create a **Override Parameters (BGPARM)** modeling object in the **Surface-to-Surface Gluing** dialog box or the **Edge-to-Surface Gluing** dialog box (Figure 1.23).

The parameter set provided by the **Glue Parameters – Linear Pair Overrides n** dialog box depends on the selected **Glue Formulation (GLUETYPE)**. If you select the **Normal and Tangential Springs** type, then in the **Glue Parameters – Linear Pair Overrides n** dialog box, you can set up the following parameters:

- Values for the normal and tangential components of penalty functions in **Penalty Normal Direction (PENN)** and **Penalty Tangential Direction (PENT)** parameters correspondingly.
- In the **Penalty Factor Units (PENTYPE)** box, you can enter the unit of measurement for penalty functions:
 - a. **1/Length** (default) is analogous to the linear contact parameter.
 - b. **Force/(Length x Area)** is similar to the linear contact parameter.

If you select the **Weld Like Connection** type, the **Glue Parameters – Linear Pair Overrides n** dialog box provides the following parameters:

- The penalty function value for the **Penalty Factor (PENGLUE)** parameter.
- In the **Penalty Factor Units (PENTYPE)** box, you can enter the unit of measurement for penalty functions:
 - a. **Unitless** (default).
 - b. **Force/(Length x Length)**.

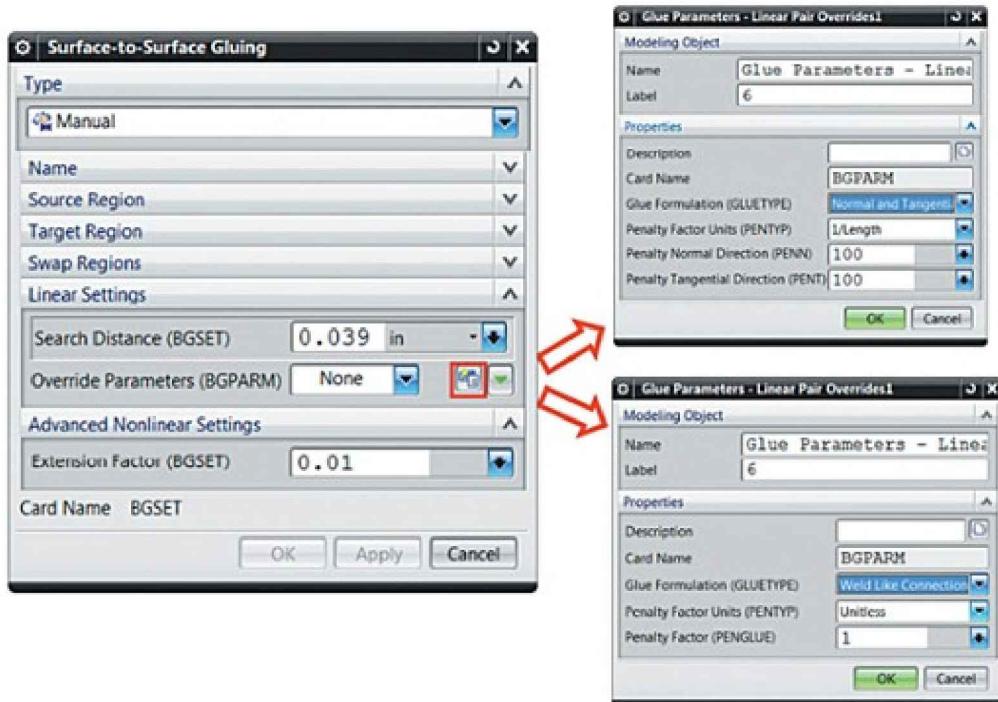


Figure 1.23. Specifying local gluing parameters

You can specify the global parameters in the *Case Control tab of the Solution* dialog box, which is described in detail in para. 1.4.4 of this chapter. To specify global gluing parameters, use the **Create Modeling Object** command for *Global Glue Parameters* to open the **Glue Parameters – Linear Global n** dialog box (Figure 1.24). Note that this dialog box contains the same parameters as the local parameters dialog box. The *INTORD* and *REFINE* have the same meaning as the corresponding linear contact parameters. *INTORD* and *REFINE* parameters are described in para. 1.5.2 of this chapter.

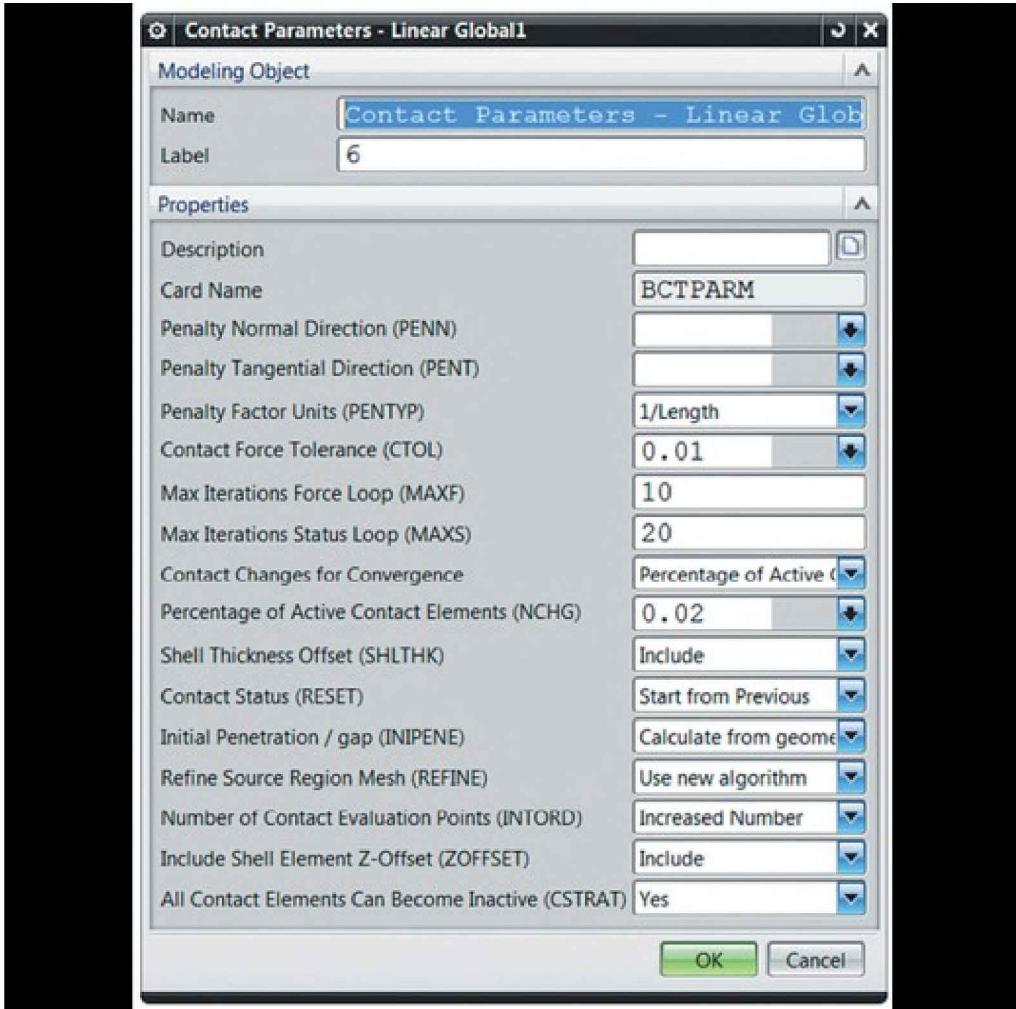


Figure 1.24. Specifying global gluing parameters

1.6.2. Gluing results output

The following values can be output for the gluing connection types (Figure 1.25):

- *Glue Pressure – Nodal*
- *Glue Force – Nodal*
- *Glue Traction – Nodal*

To output these results, in the **Solution** dialog box, in the **Case Control** tab, use the **Output Requests** list. Either change the *Modeling Object currently used*, or create a new one. In the **Structural Output Requests n** dialog box, in the **Glue Result** tab, select the *Enable BGRESULTS Request* check

box. Then select values to output into the *.op2 results file for viewing in the postprocessor (Figure 1.25).

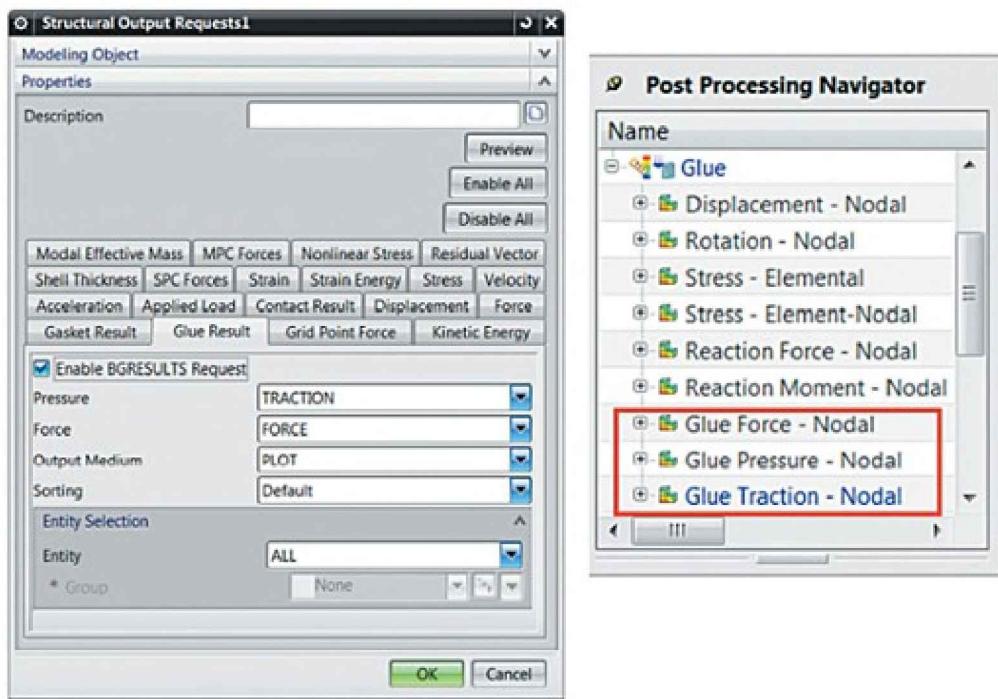


Figure 1.25. Writing gluing results into a *.op2 results file

1.7. Thermo elastic analysis

To analyze the stress-strain state of structures under variable temperature conditions such as non-uniform temperature, heat flux, heat generation, radiation, you need to account for temperature strain and stresses. Such problems of deformable solid body mechanics belong to the class of thermo elastic problems.

NX Advanced Simulation can solve thermoelastic problems in one of three ways:

1. Direct entry of a temperature distribution from a thermal conductivity problem solved earlier.
2. Specifying a thermal analysis input file as the initial temperature load.
3. Using NX Multi-Physics Solution module for sequential or linked solution of static and temperature analysis.

Since the first two methods use a sequential combination of temperature and static analyses, they are discussed in this section using a steady-state thermal conductivity problem as an example.

NX Multi-Physics Solution application interface is discussed separately in section 1.7.2 of this chapter.

1.7.1. Example of thermo elastic problem solution

Consider a hollow thick-shell steel sphere with a diameter of 10 cm and wall thickness of 7 cm. A temperature of 100 °C is maintained inside the sphere, and a temperature of 20 °C is maintained outside the sphere (Figure 1.26, A). To take advantage of symmetry, the FE model is constructed for one eighth of the sphere (Figure 1.26, B). To solve the problem, you must find strain and stress caused by non-uniform temperature expansion. It is assumed that temperature analysis results will then be used as temperature loads in static analysis. The distribution of the temperature field in the sphere can be found by solving a steady-state thermal conductivity problem. This problem can be solved by performing the following steps:

- Solving the thermal conductivity problem.
- Saving thermal conductivity analysis results for the first and the second methods.
- Accounting for the temperature distribution in the static problem using the first and the second methods.

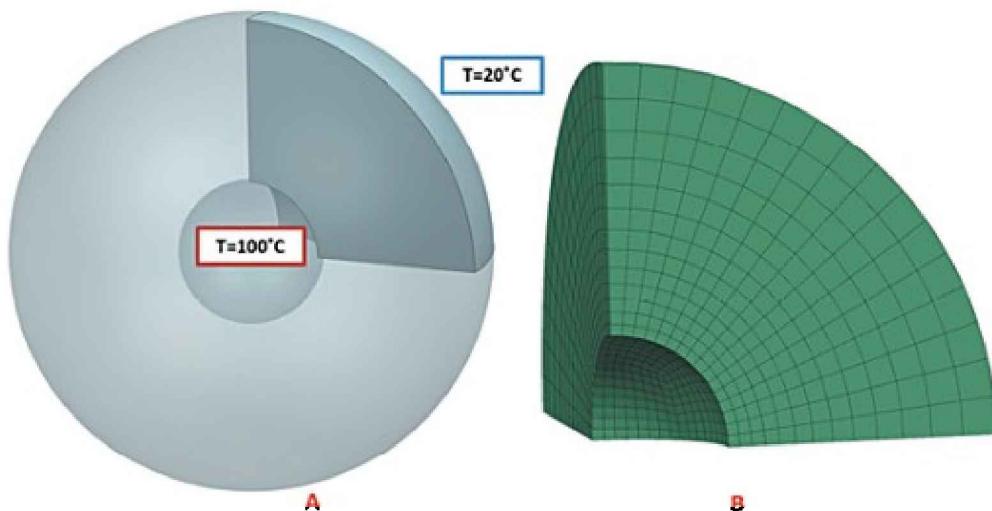


Figure 1.26. Model of a sphere:

A. Temperature boundary constraints. B. FE model with symmetry taken into account

To create a new solution of the steady-state thermal conductivity problem, right-click the simulation file in the **Simulation Navigator** tab, choose the **New Solution** command, and in the **Solution** dialog box, set up the following options (Figure 1.27):

- Under **Analysis Type**, select *Thermal*.
- Under **Solution Type**, you can select one of the two solution types:
 - a. *NLSCSH 153* corresponds to the standard thermal conductivity problem.
 - b. *NLTCSH 159* corresponds to the unsteady-state thermal conductivity problems.

Select the *NLSCSH 153* solution type.

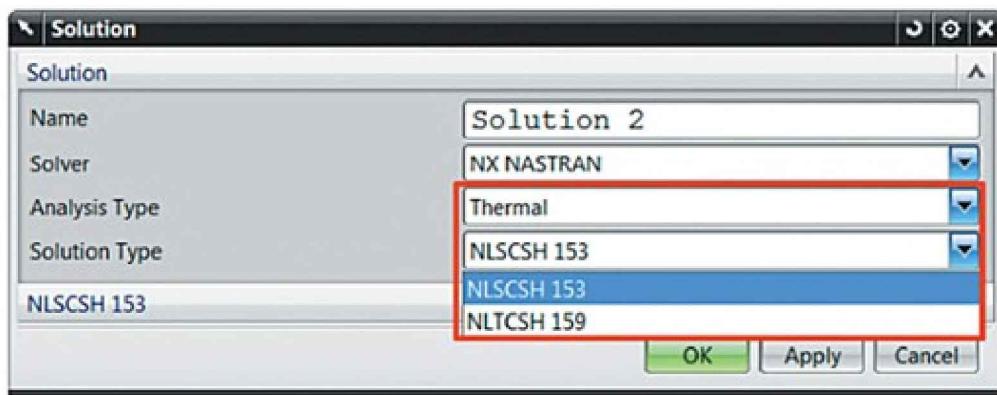


Figure 1.27. Specifying solution type for the thermal conductivity problem.

In the **Simulation Navigator** tab, in the **Advanced Simulation** toolbar, use the **Thermal Constraints** command in the **Constraint Type** command set. In the dialog box, specify temperature boundary constraints: 100 °C for the interior face, and 20 °C for the exterior face. By default, a zero heat flux condition is applied to all other faces.

To solve the steady-state thermal conductivity problem, you need to specify the thermal conductivity coefficient in addition to the material properties.

Figure 1.28 shows solution results, that is, the temperature distribution in the

sphere.

Temperature - Nodal, Scalar
Min : 20.00, Max : 100.00, Units = C

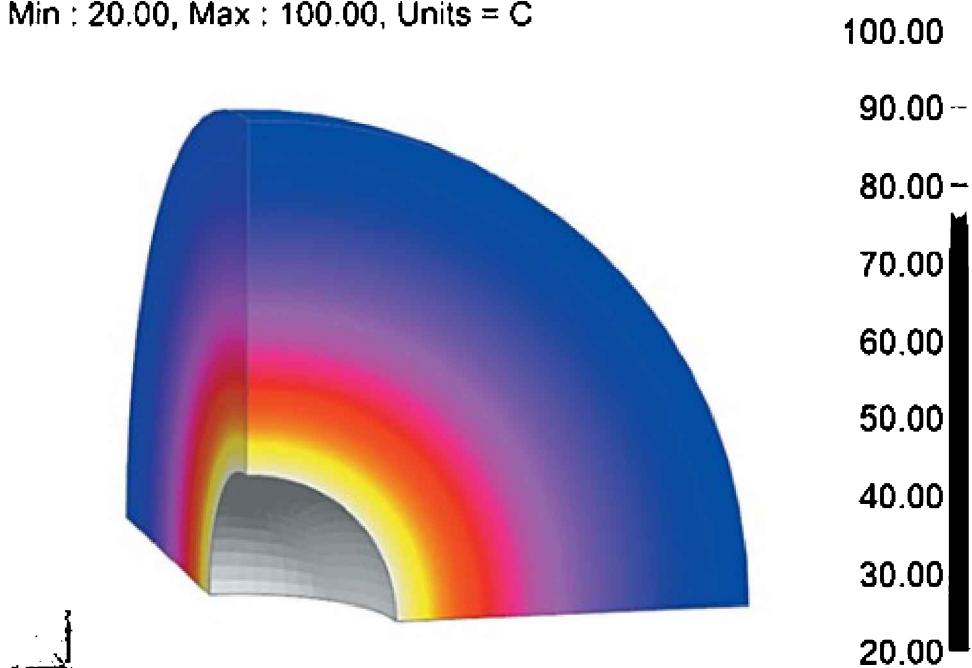


Figure 1.28. Temperature distribution field

Now you need to save the thermal conductivity results into a file.

For the first solution method of the thermoelastic problem, save temperature results for nodes or coordinates.

When the temperature distribution is displayed, in the **Post Processing Navigator** tab, use the **Identify Results** command of the Post Processing toolbar (refer to par. 5.4 of Part 1). Select all nodes of the structure in question and **Export Selection to a File**. The created file is used later in the **Import From File** command.

When exporting data, it is important to note the particular way the temperature loads are applied in the linear static problem. If the FE model used for the thermal conductivity problem is the same as the FE model used for the linear static problem, you can save temperature data for nodes. If different FE models are used, the temperature distribution results must be saved for coordinates.

In the **Identify File Export** dialog box, in the **Column Control** group, select components that will be used in the subsequent import operation.

For the second method of solving the thermoelastic problem, use a *.op2 file to specify temperature loading.

To find the stress-strain state of the sphere with temperature loading, you need to solve the linear static problem. The same FE model of one-eighth of the sphere (Figure 1.26, A) is used as for temperature calculation. A symmetry constraint is applied to flat faces of the model. When you solve linear static problems with temperature deformations, you need to specify the thermal expansion coefficient in the material properties.

If the first method is used, you can specify thermal loading using the **Temperature Load** command from the **Load Type** command set on the Advanced Simulation toolbar (Figure 1.29).

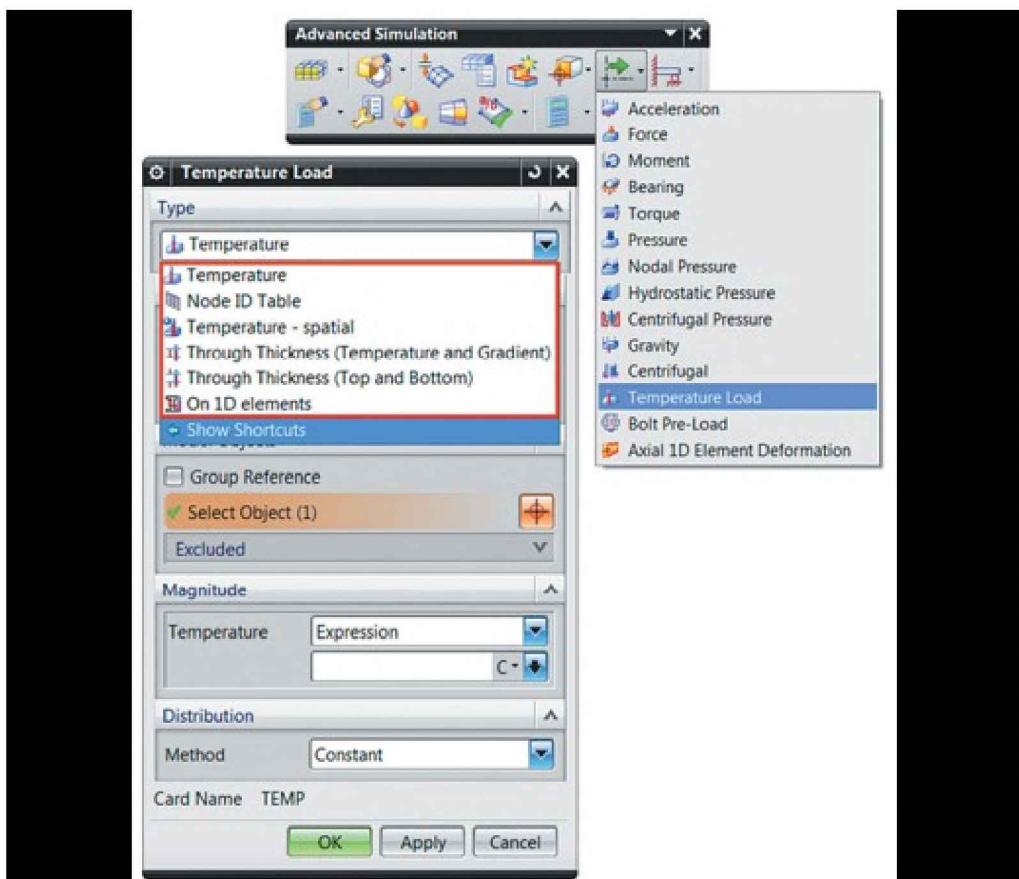


Figure 1.29. Defining temperature loading

In the *Type* group of the dialog box, you can specify thermal loading by selecting a file with distribution data. The process of specifying a data distribution field when setting up loading is described in detail in par. 4.3 of Part 1. For the current example, a more convenient option is the *Node ID Table* type because the same FE model is used for the temperature problem and the linear static problem.

If you use the second method, select the *.op2 result file obtained by solving the thermal conductivity problem.

In the shortcut menu of the simulation case of the linear static problem solution, choose *Edit* and in the *Solution Step* dialog box, in the *Temperature Pre-Load* group, for the *Pre-Load Type* option, select the solver whose results you want to use to specify the pre-load (Figure 1.30).

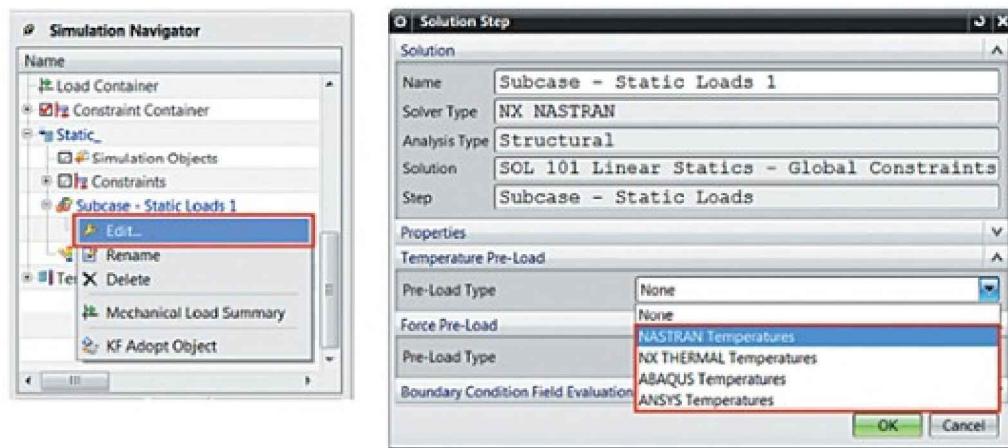


Figure 1.30. Specifying temperature loading by selecting a *.op2 results file

You can use solution results of the thermal conductivity problem with the following solvers:

- *NASTRAN Temperatures*
- *NX THERMAL Temperatures*
- *Temperatures ABAQUS*
- *Temperatures ANSYS*.

It is important to note that if you use *NX THERMAL Temperatures*, you can solve the thermal conductivity problem for a FE model that is not identical to the FE model used in the linear static problem. For all other solvers (if you

use the temperature analysis file directly), you have to use identical FE models for the thermal conductivity problem and the linear static problem.

The results of solving the thermoelastic problem obtained from the first and the second methods are identical. You can see them in Figure 1.31.

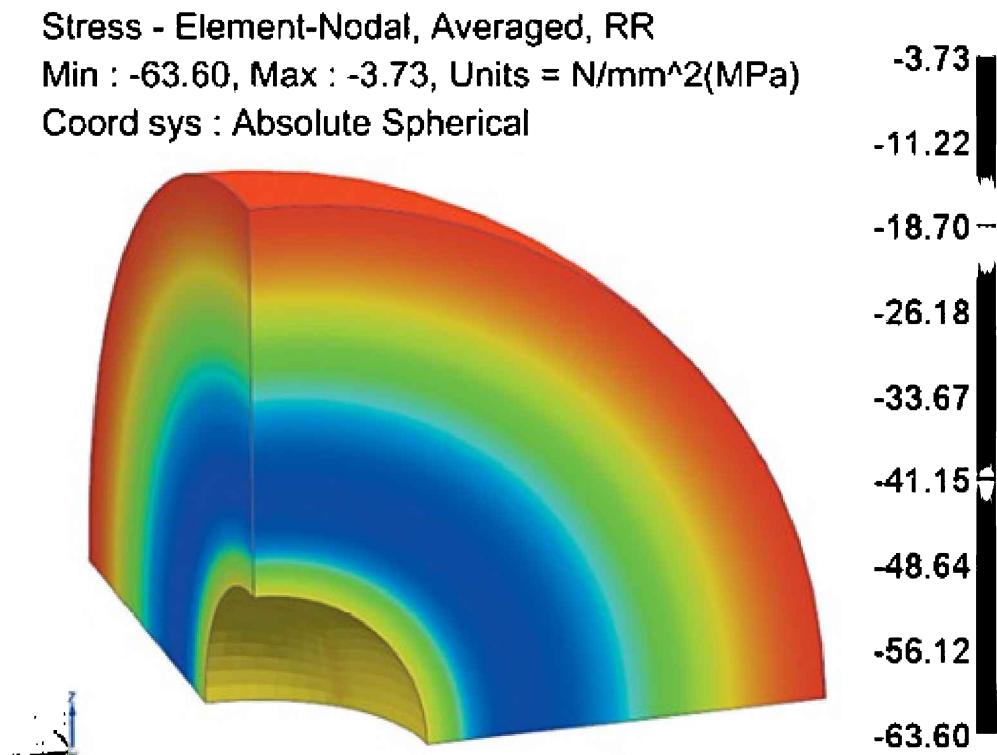


Figure 1.31. Results of solving the thermal elastic problem, radial stress distribution field, MPa

1.7.2. NX Multi-Physics Solution application

NX Multi-Physics Solution is a multidisciplinary problem solving application developed in NX Open environment. In NX 8, you can solve problems while accounting for equations of the temperature problem and linear static equations sequentially or simultaneously. Solutions with simultaneous consideration of several process physics are necessary if the formulation of the linear static problem depends on results of temperature analysis and vice versa. For example, if the surface contact zone depends on thermal expansion of bodies, and heat transfer depends on the area of contact.

To launch the application, add the corresponding command button to the

toolbar. When you use this command, the ***Multi-Physics Solution*** dialog box opens, containing four tabs (Figure 1.32).

The **MP** tab contains the following parameters:

- In the *Solution Name* box, type the name of the multi-physics solution.
- In the *Coupling Mode* list, select one of the two options:
 - a. *Iterative* to determine the solution in several steps of static and temperature analysis.
 - b. *Sequential* to determine the sequential solution of static and temperature analysis in a single step.
- In the *Dynamics Type* list, select:
 - a. *Steady State* to solve the steady-state thermal conductivity problem.
 - b. *Transient* to solve the unsteady-state thermal conductivity problem.

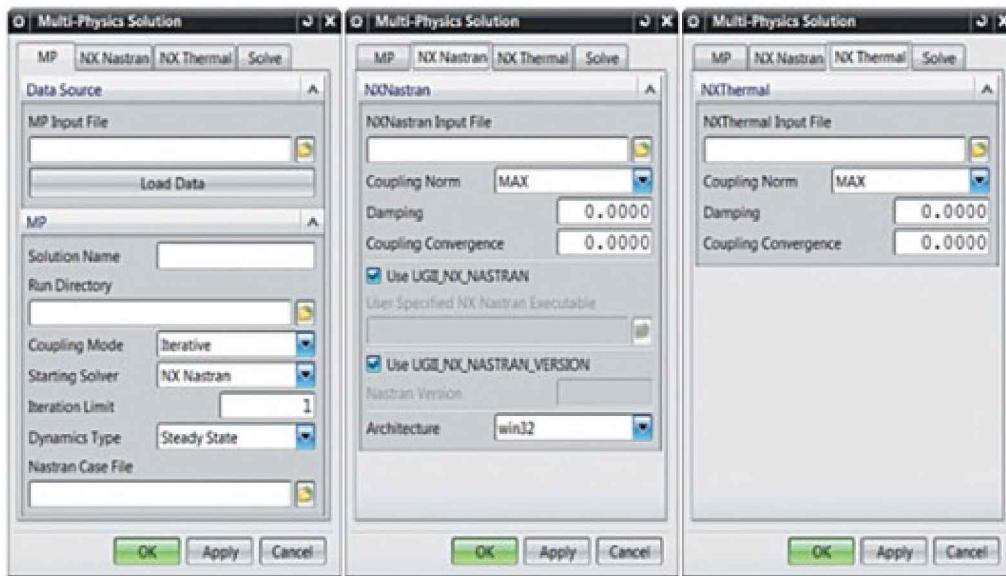


Figure 1.32. NX Multi-Physics Solution interface

In the *NX Nastran* tab, in the *NXNastran Input File* box, specify the prepared *.dat input file of the linear static problem.

In *NX Thermal* tab, in the *NXThermal Input File* box, specify the prepared *.dat input file of temperature analysis.

When all data are specified, click *Solve* to run the simulation.

1.8. Optimization analysis

Creation of a new product often involves finding its optimal design. The optimization process varies parameters of the structure and produces multiple alternative design variants from which you can select the best. Practical optimization is a process of iteratively reaching the optimum combination of design parameters. In terms of structural optimization in NX Nastran, there are three types of project parameters:

- The objective function is a parameter computed at each iteration. It characterizes the main goal of the optimization, for example, minimized mass, increased stiffness, and alleviated stress in the structure.
- Design variables are a set of structural parameters that can be varied to reach optimization goals. You also specify the variation range of each design variable.
- Design constraints are user-defined limits on physical values of the results. You can use stress and displacement values, mass and volume, natural frequencies of the structure, and other values as design constraints. During the optimization process, the obtained physical variable values are compared to design constraints. If a value calculated at an iteration does not satisfy the specified conditions, the search for the optimal solution is continued from the previous iteration.

The optimization model can contain tens or hundreds of design variables so finding their optimal combination within design constraints while satisfying objective functions that can also be numerous, is a difficult task. In engineering practice, optimization problems are frequently solved using numerical analysis tools.

NX Advanced Simulation includes the following optimization solutions:

- Geometry optimization
- Parametric optimization
- Topology optimization
- Structural sensitivity study

This chapter provides a detailed treatment of geometry optimization as implemented in the **Geometry Optimization** module.

1.8.1. Geometry optimization

You can use NX Advanced Simulation to optimize a structure's geometry, in other words, optimize its shape. You can do this using the **Geometry Optimization** module, which supports the following NX Nastran solution types:

- Linear static analysis (*SESTATIC 101*).
- Frequency and mode analysis (*SEMODES 103*).
- Buckling analysis (*SEBUCKL 105*).
- Nonlinear static analysis (*NLSTATIC 106*).
- Steady-state thermal conductivity analysis (*NLSCSH 153*).

The optimization algorithm implemented in NX Nastran belongs to a class of methods called gradient methods [26]. The search process can be briefly described as follows: gradients of the objective function and constraints are determined at a particular point in design parameter space. These gradients then determine the search direction. The computation continues in that direction until an extreme point is found where design constraints are not violated. This point is then checked for optimality. If the point is not optimal, the process is started again and repeated until a point is reached where no further advancement towards the goal is possible without violating constraints.

You can launch the **Geometry Optimization** module in one of two ways:

- In the shortcut menu of the simulation file in the **Simulation Navigator tab**: **New Solution Process***Geometry Optimization* (Figure 1.33).
- In the main menu: **Insert**→**Geometry Optimization**.
- In either case the **Create Geometry Optimization Solution** opens, where you need to select the solution to use for geometry optimization (Figure 1.33), and the optimization type:
- Select *Altair HyperOpt* to perform geometry optimization as such.
- Select *Global Sensitivity* to carry out a sensitivity study of the structure. Its objective is determining design variables that

influence the objective functions of the analysed structure to the greatest degree within the specified design constraints. In terms of **NX Advanced Simulation**, the obtained values are called structure sensitivity coefficients. This analysis is used as a separate solution that produces results you can use as a preliminary stage of geometry optimization.

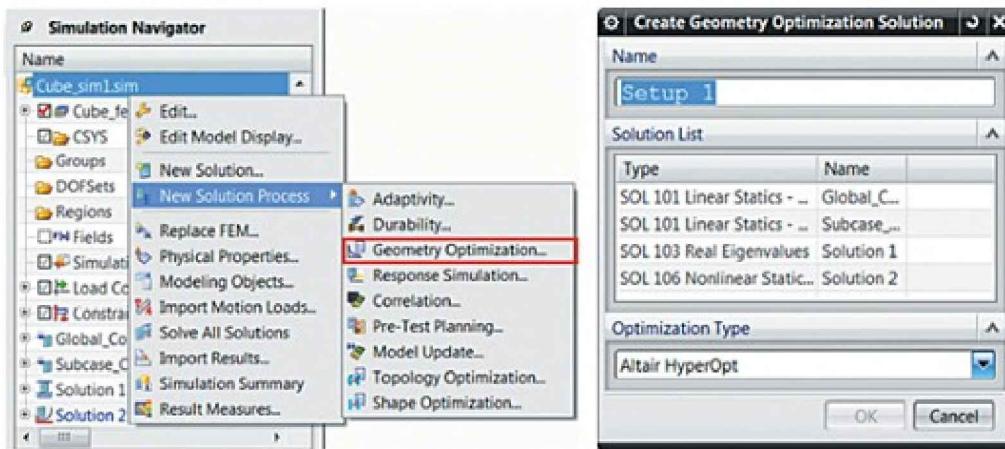


Figure 1.33. Running the **Geometry Optimization** module

Click **OK** to open the **Geometry Optimization** dialog box, which has five tabs. With the exception of the first tab, **General Setup**, where you can assign a name to the current optimization solution and select an optimization type, the rest of the tabs contain optimization parameters. The following is a description of the step-by-step process of creating the optimization solution with setting up optimization parameters in tabs of the **Geometry Optimization** dialog box.

Select the **Define Objective** tab in the **Geometry Optimization** tab and specify the objective function with corresponding parameters (Figure 1.34):

- Under **Category**, select the type of FEs participating in the optimization: **1D Objectives** to have only 1D FE meshes participate, **2D Objectives** to have 2D FE meshes participate, **3D Objectives** to have only 3D FE meshes participate, or **Model Objectives** to allow the participation of all FE meshes. If you select **Model Objectives**, the **Apply to** option is not available.
- In the **Type** list, select physical variable for optimization such as stress, displacement, strain, natural frequency, mass, volume, and

temperature.

- In the *Apply to* list, select the geometry object of the model that need optimization. The set of available geometry objects depends on the specified physical value. For example, mass and volume optimization is only available for solid bodies, whereas for stress optimization you can specify solid bodies, faces, edges, and points. If you specify a vector or tensor physical variable, you must select the relevant component.
- In the *Parameters* group, you can select minimization, maximization, or specific target value of the objective function.

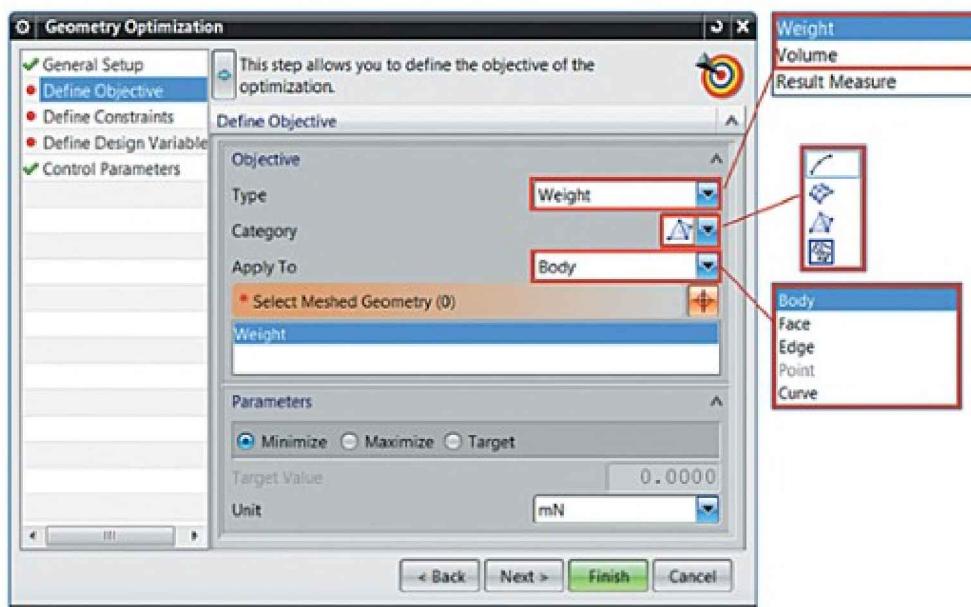


Figure 1.34. Setting objective functions

In NX 8.5 select the *Define Objective* tab in the *Geometry Optimization* tab and you can define the objective function via *Result Measure* as shown on Figure 1.35.

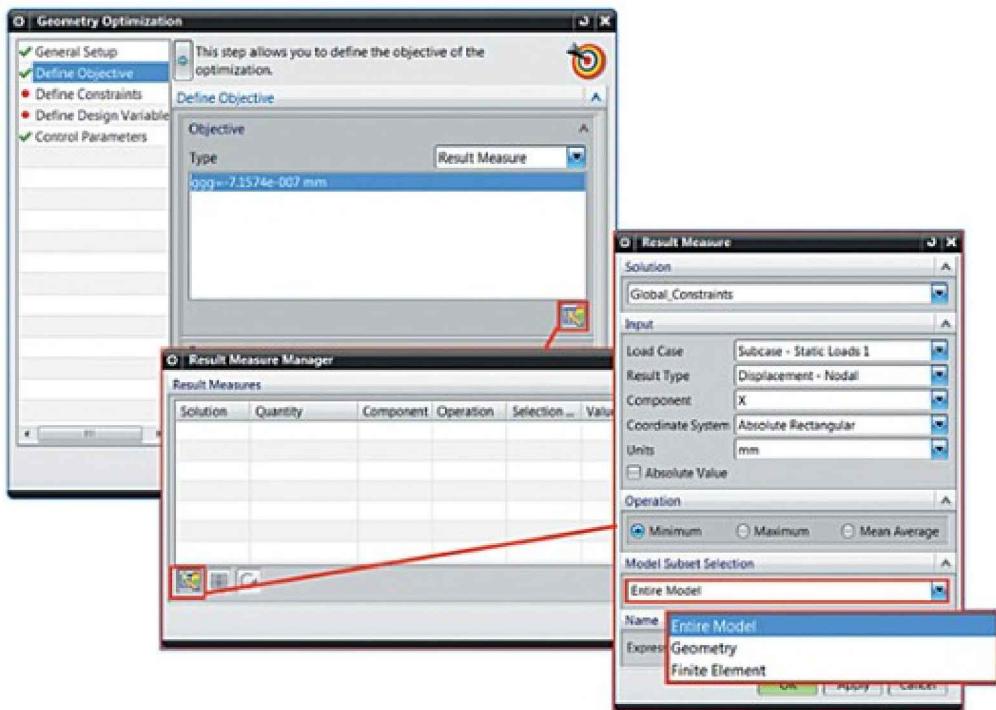


Figure 1.35. Setting objective functions via Result Measure

You can use displacement, stress and strain, reaction force, natural frequency, mass, and volume as design constraints. To open the **Define Constraints** dialog box, use the **Create Constraints** command in the **Geometry Optimization** dialog box, in the **Define Constraints** tab (Figure 1.36). To set design constraints, follow a similar procedure to specifying the objective function: select element type (*Category*), the physical variable (*Type*), and the geometry object of the model (*Apply to*). In the *Parameters* group, specify the *Limit Value* and select its type: *Upper* or *Lower* limit correspondingly.

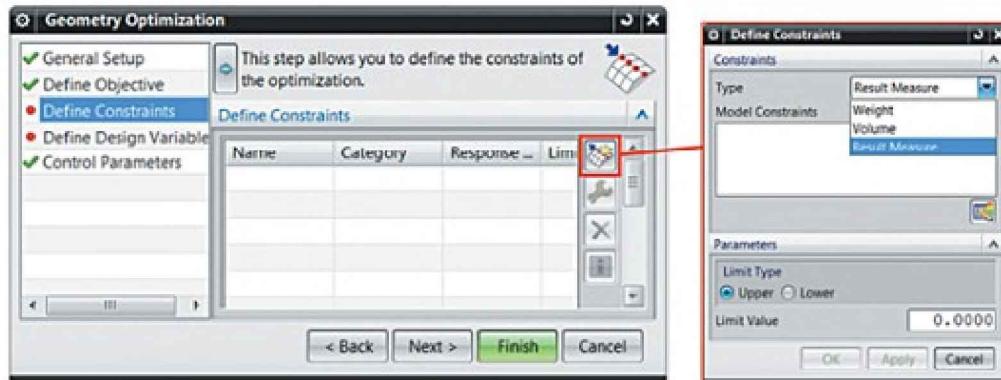


Figure 1.36. Setting design constraints

Design variables are varied to gain the optimum of the objective function within the given design constraints. You can use the following model parameters as design variables (Figure 1.37):

- Cross section parameters for 1D rod elements.
- Thicknesses of 2D shell elements.
- Geometry parameters of the CAD model.
- Sketch dimensions used to create the FE model.

Open the **Define Design Variables** dialog box by selecting the **Define Design Variables** tab and using the **Create Design Variables** command. You can select all types of design variables in the drop-down list. In the **Parameters** group, you can specify the variation range of the design variables.

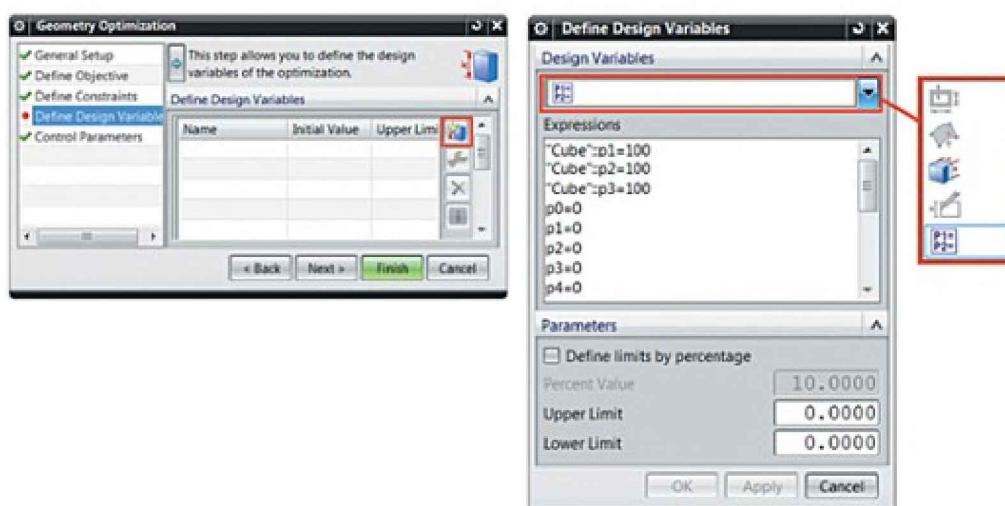


Figure 1.37. Specifying design variables

If you select **Control Parameters** in the **Geometry Optimization** dialog box, you can specify parameters for solving the optimization algorithm (Figure 1.38):

- In the *Maximum Number of Iterations* box, type the number of iterations after which the computation is stopped regardless of the result.
- In the *Maximum Constraint Violation*, type the percentage value that determines the extent to which design constraints can be violated to achieve a result.
- In the *Relative Convergence (%)* box, type the change of the objective function value (percent) over the last two iterations. The solution is considered converged if the change is less than this value.
- In the *Absolute Convergence* box, type the actual change of the objective function value over two sequential iterations. The solution is considered converged if the change is less than the value specified in this box.
- In the *Perturbation Fraction* box, enter the allowable change (as a fraction) of design variables over the first iterations.
- Select the *Save results for all iterations* check box to choose whether to save results for individual iterations.

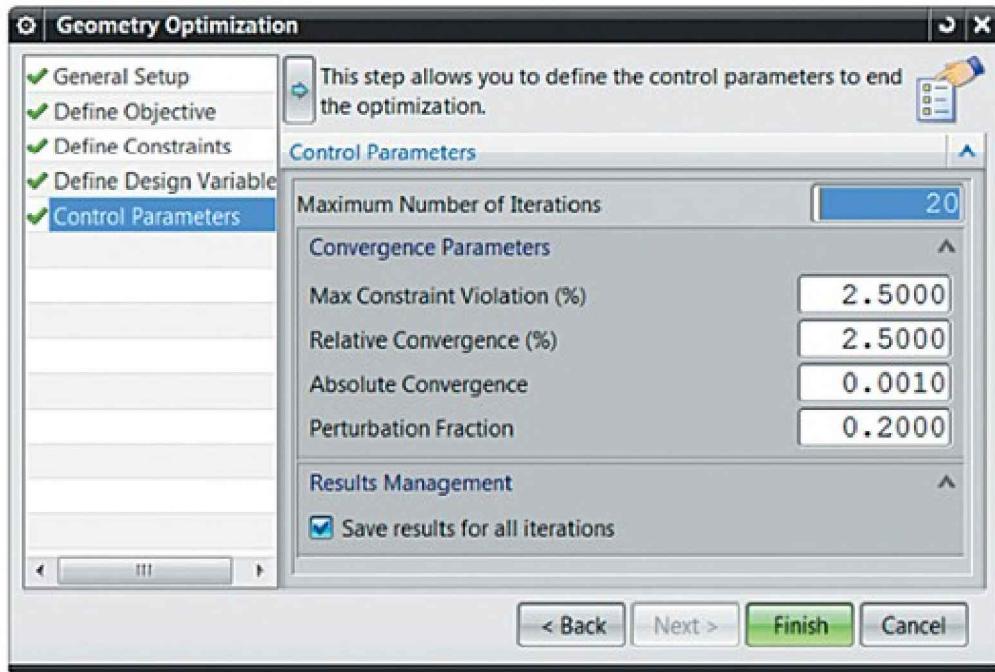


Figure 1.38. Setting optimization solver parameters

1.8.2. Other solutions for structure optimization

You can use the *DESOPT 200* solution type of NX Nastran solver to optimize the structure by varying physical properties of the simulation model.

You can use several different solution types in the same optimization process. This allows considering a broad spectrum of variables and constraints.

The following solution types are available for NX Nastran *DESOPT 200*:

- Linear static (*SESTATIC 101*).
- Natural frequency and mode analysis (*SEMODES 103*).
- Buckling analysis (*SEBUCKL 105*).
- Frequency analyses (*SEDCEIG 107*, *SEDFREQ 108*, *SEMSEIG 110*, *SEMFREQ 111*).
- Modal transition analysis (*SEMTRAN 112*).

The selection of objective functions depends on the solution type.

Analysis type	Objective functions

Linear static (SESTATIC 101)	Volume, stress/strain, displacements, reaction force
Modal (SEMODES 103)	Volume, mode number
Buckling (SEBUCKL 105)	Number of buckling mode
Frequency	Dynamic strains, displacement, velocity, acceleration, frequency values
Modal transition (SEMTRAN 112)	Dynamic strains, displacement, velocity, acceleration, frequency values

You can use the following design variables:

- Material properties (elastic modulus, shear modulus, Poisson ratio, material density, temperature expansion coefficient).
- Physical properties of 2D elements, including multi-layer shells (ply thicknesses, material orientation).
- Properties of 1D and 0D elements (section parameters, inertial and stiffness characteristics, damping).
- Node location of 2D, 1D, and 0D elements.

1.9. Example. Solving a problem with linear contact interaction

This example demonstrates a solution of a thermal conductivity problem with contact interaction using a prepared FE model. The structure consists of a thin-shell cylinder clamped between flat sheets (Figure 1.39).

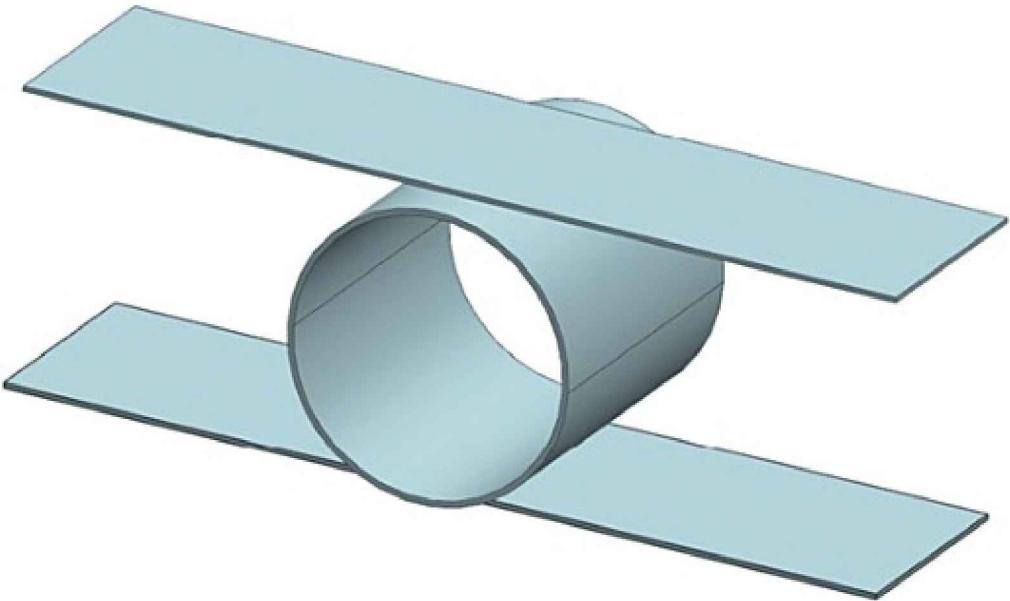


Figure 1.39. A thin-shell cylinder between flat sheets

It is assumed that the lateral edges of plates are constrained as shown in Figure 1.40, and are maintained at a temperature of $100\text{ }^{\circ}\text{C}$, while a temperature of $500\text{ }^{\circ}\text{C}$ is maintained on the lateral edges of the cylinder.

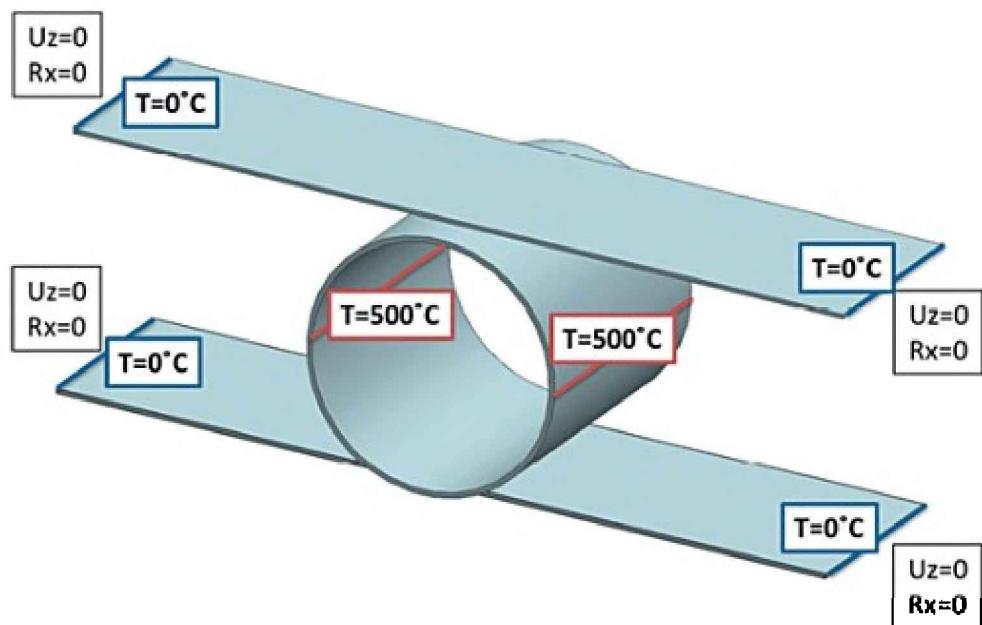


Figure 1.40. Temperatures and constraints of the structure

The FE model was prepared using midsurfaces constructed from thin-shell bodies with symmetry. The structure, the boundary conditions, and the loads are symmetrical with regard to XOY, XOZ, and YOZ planes (Figure 1.41). Thicknesses of 2D shell elements are automatically inherited from thicknesses of geometry. The material of all bodies is Steel from NX Advanced Simulation materials library.

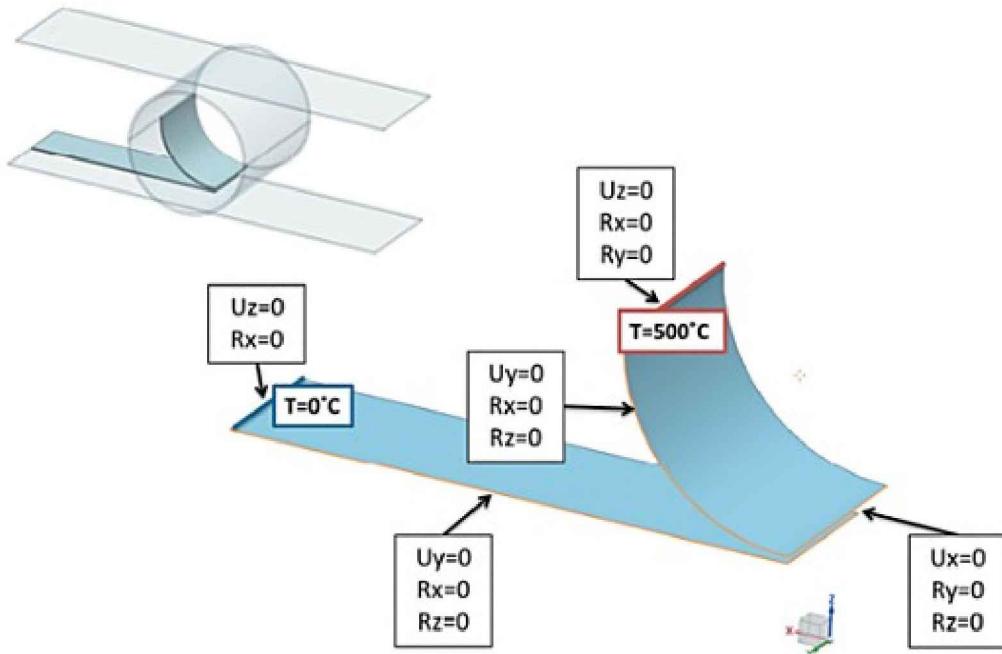


Figure 1.41. Shell model of the structure with symmetry

In the thermal conductivity problem solution, the touching surfaces are “glued”, while for the purposes of linear static analysis, they are in linear contact.

Engineering analysis of this problem is performed over several steps:

1. Opening the FE model and creating a new simulation model file.
2. Formulating the temperature analysis problem: setting gluing conditions, temperature loads, and boundary constraints with symmetry.
3. Performing the temperature analysis.
4. Creating a linear static analysis solution
5. Specifying boundary conditions, temperature distribution, and linear contact.
6. Performing linear static analysis and viewing the results.

In this example, you use commands on the **Advanced Simulation** toolbar.

Opening the FE model and creating a new simulation model file

Launch NX and open the *Cylinder_fem.fem*, file. Go to the NX Advanced Simulation module: *Start→Advanced Simulation*.

Reset all dialog box settings to default, and in the main menu, choose: **Preferences→User Interface→General→Reset Dialog Box Settings**, then click **OK**.

Right-click the FE model in **Simulation Navigator** or the **Simulation File View** dialog box. Choose **New Simulation** to create the simulation model. In the simulation file creation dialog box, enter the name of the new file (*Cylinder_sim1*) and a folder to save it. Make sure you select NX Nastran as the template, then click **OK**. In the simulation file creation dialog box, click **OK**. In the new **Solution** dialog box, select the solver and the new solution:

- In the **Name** box, type *Thermal_cyl*.
- Under **Analysis Type**, select *Thermal*.
- Under **Solution Type**, select *NLSCSH153*.

Click **OK**.

Thus the simulation model is created from the source FE model. In the **Simulation File View**, the simulation model file is active. Note that the model tree of the simulation file is populated with empty containers that are filled when the actual simulation model is created.

Formulating the temperature analysis problem: setting gluing conditions, temperature loads, and boundary constraints with symmetry

The example FE model contains 2D shell finite elements that are not connected to each other. To describe the thermal connection between them correctly, a **Surface-to-Surface Gluing** simulation model is created. To create it, in the **Simulation Object Type** list on the Advanced Simulation toolbar, choose **Surface-to-Surface Gluing**.

In the dialog box, do the following (1.42):

- Select *Automatic Pairing* type and click **Create Face Pairs**.
- In the new dialog box, select all faces as shown in Figure 1.42.
- The relevant faces are separated by a distance equal to half the sum of thin-shell body thicknesses, therefore enter 4 mm in the *Distance Tolerance* box.
- Click **OK**.
- Enter 1 mm in the *Search Distance* box.
- Click **OK**.

Note that a new object corresponding to the created connection is added to the *Simulation Object Container* node of the model tree.

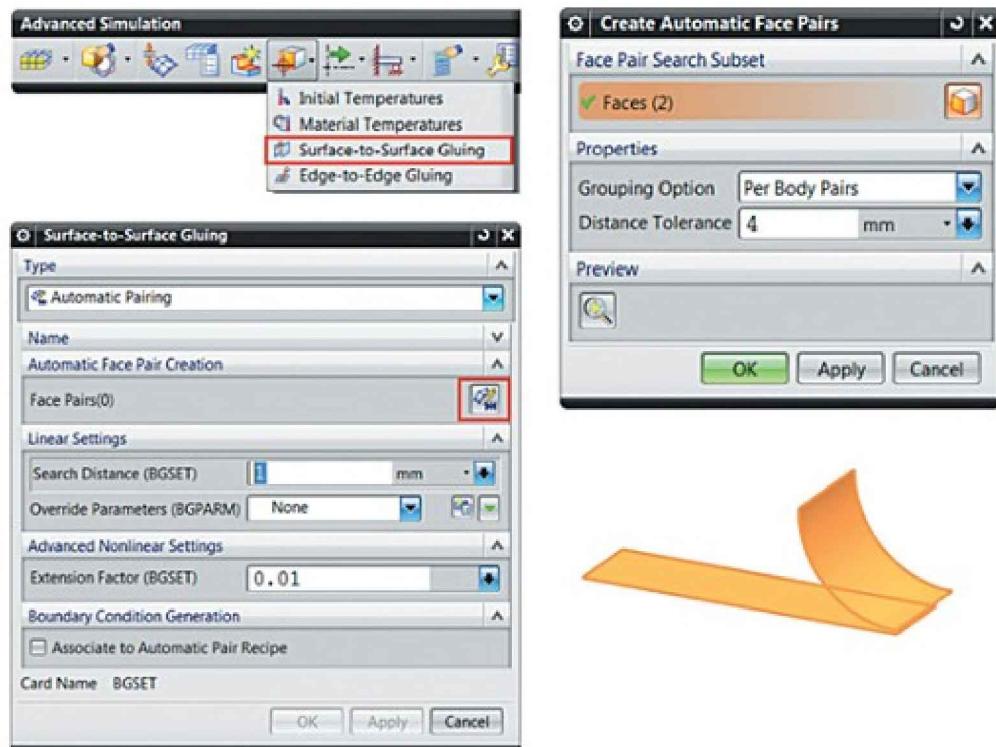


Figure 1.42. Setting up a Surface-to-Surface Gluing simulation object

To set boundary conditions, on the **Advanced Simulation** toolbar, in the **Constraint Type** list, select the **Thermal Constraint** option. In the dialog box, select the left lateral edge on the flat shell as shown in Figure 1.43 and enter 0 °C in the *Temperature* box. Click **Apply**.

Repeat the procedure for the second constraint. Now select the top edge of the cylinder surface and enter a temperature value of 500 °C. Click **OK**.

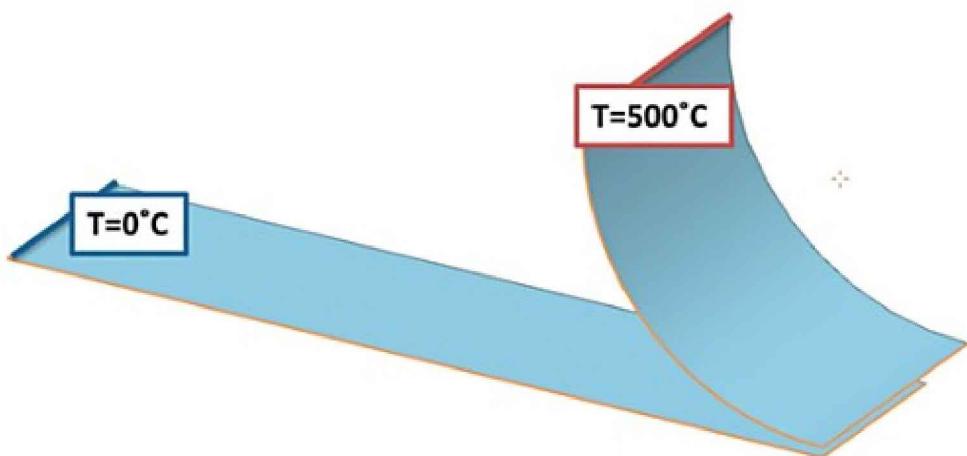


Figure 1.43. Setting up temperature boundary conditions

Temperature analysis and viewing results

In the simulation model tree in **Simulation Navigator**, you can see containers for loads, constraints, and simulation objects. Expand these and make sure they are set up correctly. The model is ready for simulation. Save the model. To do so, in **Simulation Navigator**, right-click *Cylinder_sim1* in the **Simulation File View** and choose **Save**.

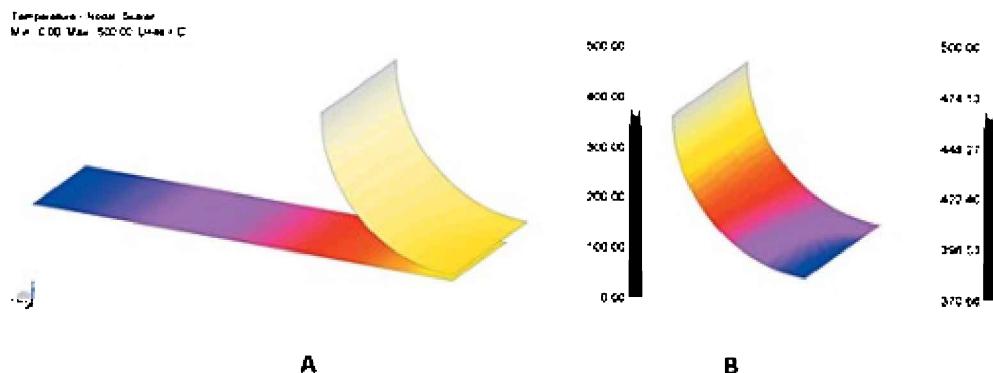
To run the solution of the simulation model, right-click *Thermal_cyl* and choose **Solve....** Click **OK**.

When you see the *Job Finished* message in **Solution Monitor**, close all new solution dialog boxes.

To load and analyse the results, in the **Simulation Navigator** simulation model tree, double-click the *Results* node of the active *Thermal_cyl* solution. To view the results, in the **Post Processing Navigator** tab, expand the *Thermal_cyl* node and double-click *Temperature – Nodal* to select it.

The graphics area displays the nodal temperature distribution (Figure 1.44, A).

Now, working independently, use commands of the **Post Processing** toolbar to display results for the cylinder surface only (you will need to create a group of corresponding elements or nodes) (Figure 1.44, B).



*Figure 1.44. Nodal temperature distribution fields:
A. For the entire structure. B. For the thin-shell cylinder*

Creating a linear static analysis solution

Create one more simulation case for the static analysis. To do so, in **Simulation Navigator**, right-click **Cylinder_sim1** and choose **New Solution**.

In the **Solution** dialog box, select the solver and the new solution:

- In the **Name** box, type **Static_cyl**
- Under **Analysis Type**, select **Structural**.
- Under **Solution Type**, select **SESTATIC 101 – One constraint**.

Click **OK**.

In the model tree of the simulation file, note that the **Static_cyl** node contains the automatically created **Subcase – Static Loads 1** simulation case.

Specifying boundary conditions, temperature distribution, and linear contact

On the **Advanced Simulation** toolbar, click **User Defined Constraint**. Specify boundary conditions for edges that correspond to constraints and symmetry conditions as shown in Figure 1.45.

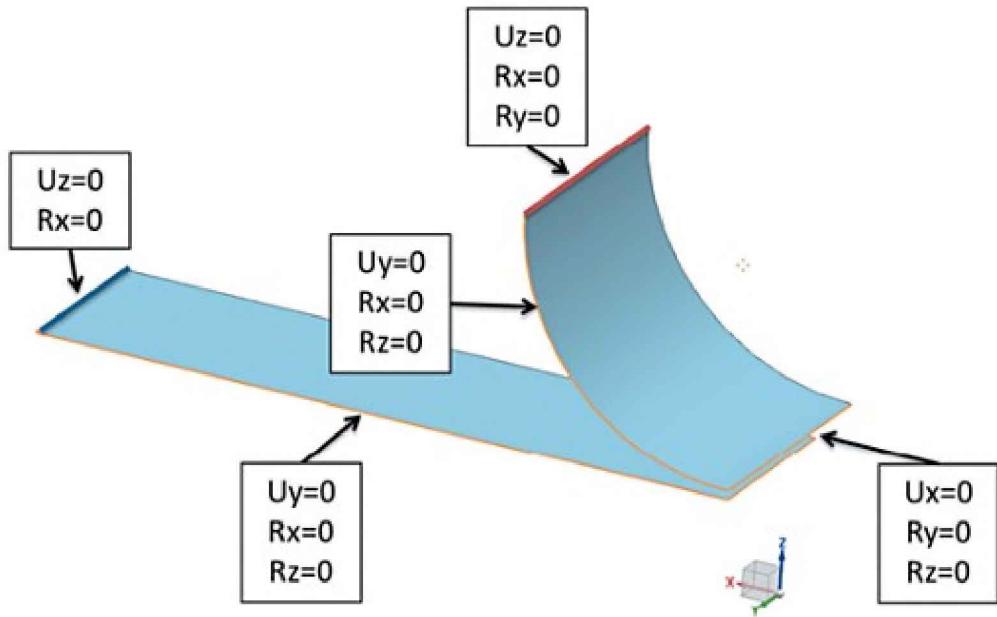


Figure 1.45. Setting up boundary conditions

A new *Update* icon on the *Static_cyl* solution node in the **Simulation Navigator** tab indicates a conflict of constraints. To resolve the conflict, right-click *Static_cyl* and choose **Resolve Constraints...** in the shortcut menu. A dialog box opens with a list of conflicts. Do the following for each conflict (Figure 1.46):

- Right-click the conflict to open the shortcut menu.
- Choose the *Keep Overlapping* command, that is, use both constraints in the relevant zones.

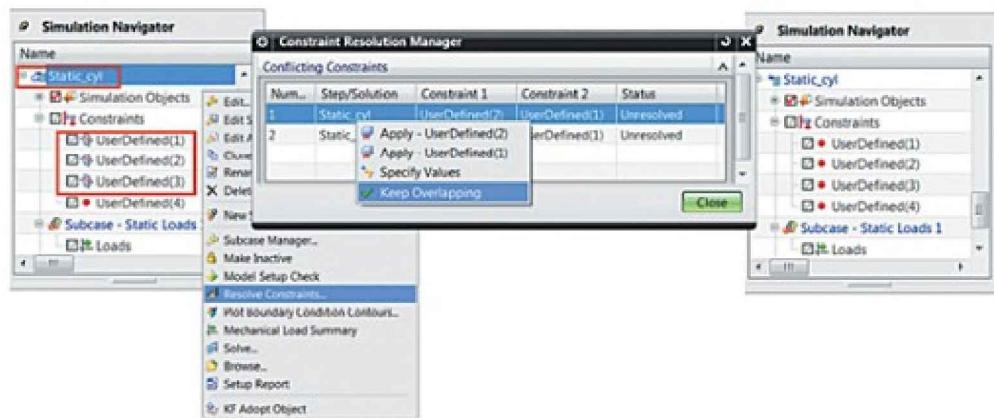


Figure 1.46. Resolving constraint conflicts

To set up linear contact of bodies, use the **Surface-to-Surface Contact** command on the **Advanced Simulation** toolbar.

In the dialog box, do the following (1.47):

- For the *Automatic Pairing* type, use the **Automatic Face Pair Creation** command.
- In the new dialog box, select all faces.
- In the *Distance Tolerance* box, enter 4 mm.
- Click **OK**.
- In the *Target Contact Side* list, select *Bottom*.
- Click **OK**.

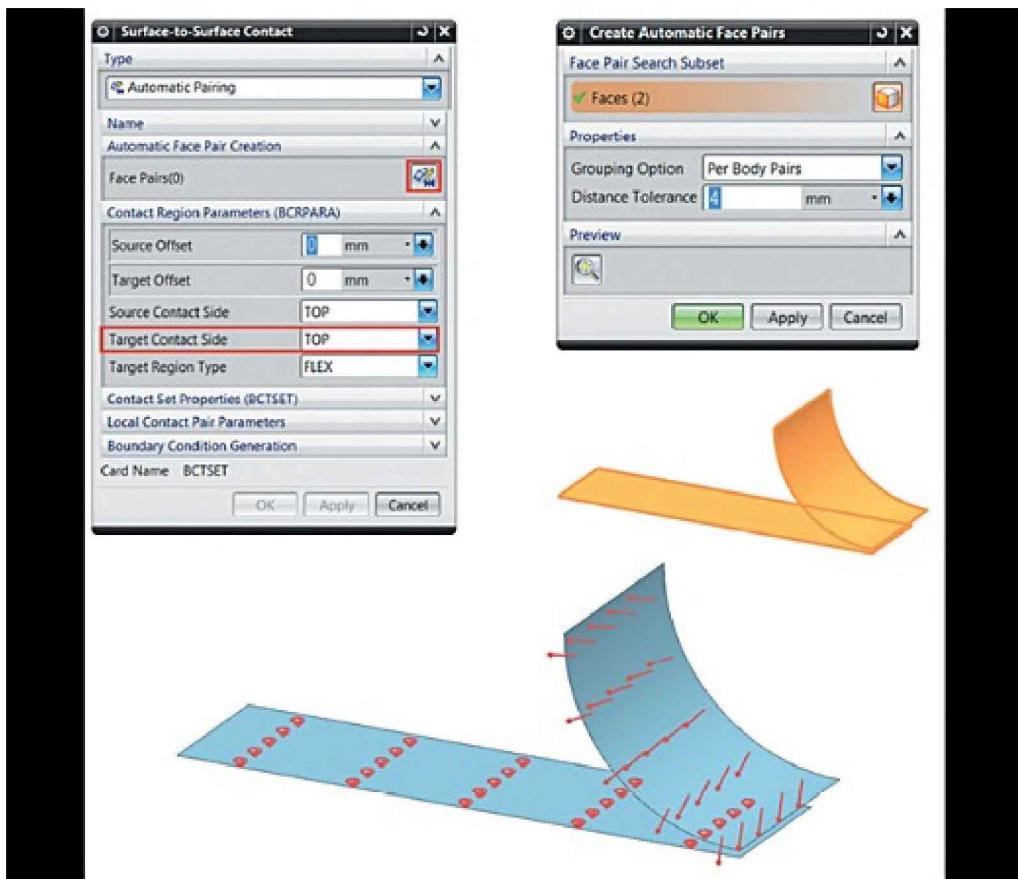


Figure 1.47. Setting up contact interaction

After you complete the previous actions, in the **Simulation Navigator** tab, in the **Simulation Object Container** of the model tree, a new object is added for the contact you created.

Since it is assumed that a temperature field acts on the model, you also need to specify the initial temperature. To do so, on the **Advanced Simulation** toolbar, use the **Initial Temperatures** command. In the dialog box, select all bodies and specify a temperature value of 20 °C.

To specify temperature loading using the thermal conductivity problem results file, in the **Simulation Navigator** tab, right-click the **Subcase – Static Loads 1** simulation case node, and choose **Edit** in the shortcut menu. In the **Solution Step** dialog box, in the *Temperature Pre-Load* group, enter the following data:

- In the *Pre-Load Type* list, select *NASTRAN Temperatures*.
- For the new *NASTRAN Pre-Load Results File* option, specify the *.op2 file with the solution of the thermal conductivity problem, that is, *Cylinder_sim1-thermal_cyl.op2*.

Running the simulation and viewing static analysis results

In the simulation model tree in **Simulation Navigator**, in the *Static_cyl* solution node, you can see containers for loads, constraints, and simulation objects. Expand these and make sure they are set up correctly. The model is now ready for simulation. Save the model. To do so, in the **Simulation Navigator** tab, right-click *Cylinder_sim1* in the **Simulation File View** and choose **Save**.

To run the solution for the created simulation model, right-click *Static_cyl* and choose **Solve....** Click **OK**.

When you see the *Job Finished* message in the **Solution Monitor** dialog box, close all new solution dialog boxes.

To load and analyse the results, in the **Simulation Navigator** simulation model tree, double-click the *Results* node of the active *Static_cyl* solution. To view the results, in the **Post Processing Navigator** tab of the Resource Bar, expand the *Static_cyl* node and double-click *Nodal Displacement* to select it.

The graphics area displays the nodal displacement field (Figure 1.48, A).

Now, working independently, use commands of the **Post Processing**

toolbar to display results for the plate only (Figure 1.48, B) (you will need to create a group of corresponding elements or nodes). The plate is deflected due to the contact interaction with the thin-shell cylinder, which expands under temperature load.

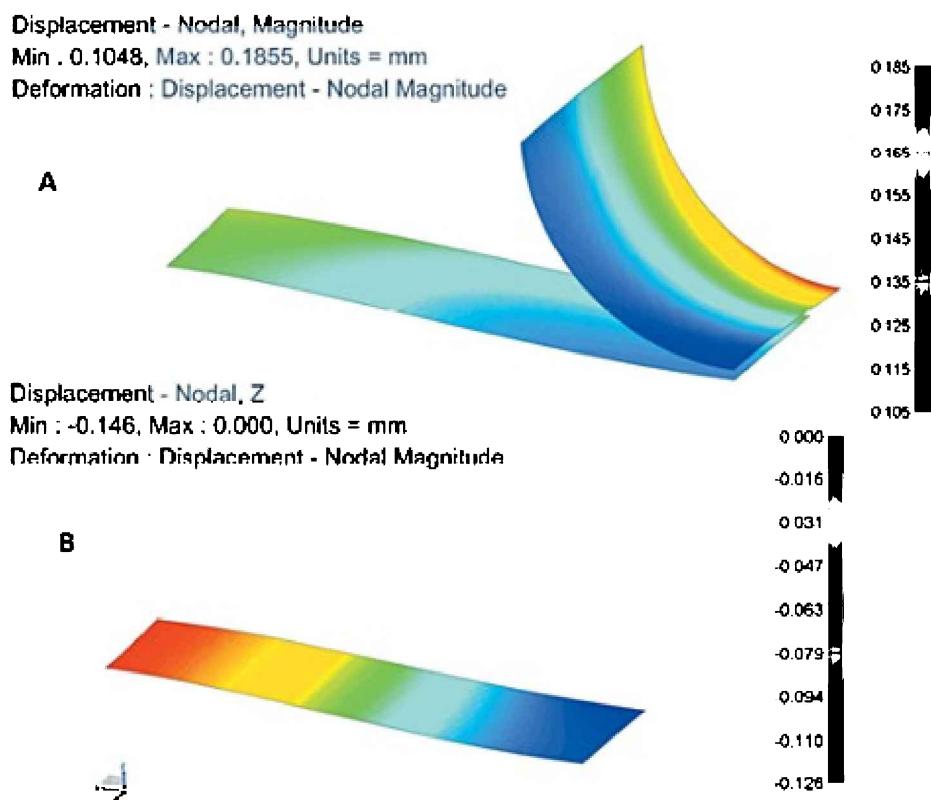


Figure 1.48. Distribution:

- A. Absolute value of the displacement module for the entire structure. B. Z-axis components of the displacement vector for the plate*

Chapter 2. Buckling analysis

Stability is a key structural performance criterion. Stability is the ability of a structure to return to the initial equilibrium state after external loading is removed. If the structure adopts a state that is not identical to the initial state, it is said to have buckled. The load necessary to cause buckling is the critical load, and states that immediately precede buckling are known as critical states. A structure's behaviour after buckling is known as post-critical behaviour.

Buckling typically occurs in structural elements that have one or two dimensions significantly smaller than the third, for example, rods, beams, or plates.

As structures transition to a post-critical state, they exhibit three types of behaviour [3]:

1. Emergence of new equilibrium states that are adjacent to the initial state.
In this case the structure typically has a single stable equilibrium state prior to the first critical state. When critical load is reached, new adjacent equilibrium states occur that are qualitatively different from the initial stable state. Figure 2.1 shows a stiff rod under central compression (the Euler problem), and its equilibrium state diagram. The event of new adjacent equilibrium states arising is called a bifurcation, and point A is called the bifurcation point.

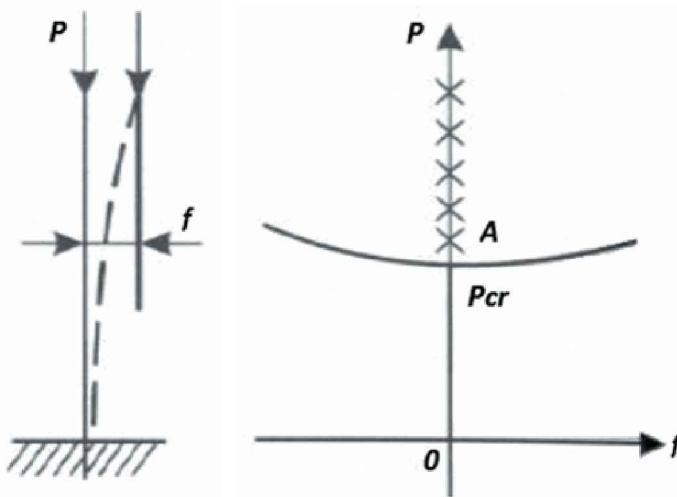


Figure 2.1. Equilibrium states adjacent to the initial state

2. Emergence of new equilibrium states that are not adjacent to the initial shape. This type of buckling can be demonstrated using the example of a membrane shown in Figure 2.2. When load q reaches a critical magnitude, displacements of membrane f increase to point B (critical point on the equilibrium curve). A subsequent slight increase of the load causes an abrupt transition from state 1 to non-adjacent state 2 (point B').

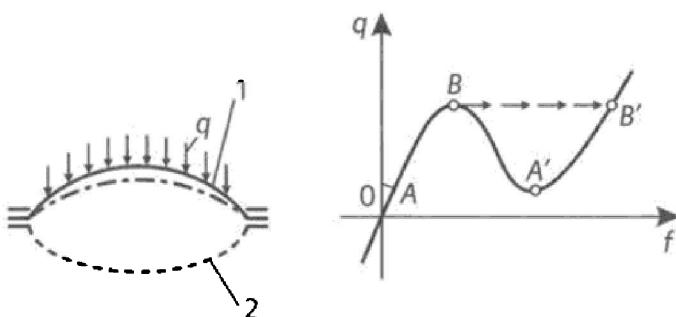


Figure 2.2. Equilibrium states not adjacent to the initial state

3. Absence of static equilibrium states in post-critical condition. Reaching critical load values eliminates static equilibrium states, and the structure is set in motion. This motion can be aperiodic with gradual departure from the initial equilibrium state or oscillatory with increasing amplitude. One example of such behaviour is a rod with a so-called follower load on its end (Figure 2.3). The direction of this load changes with the changing deformation state of the rod.

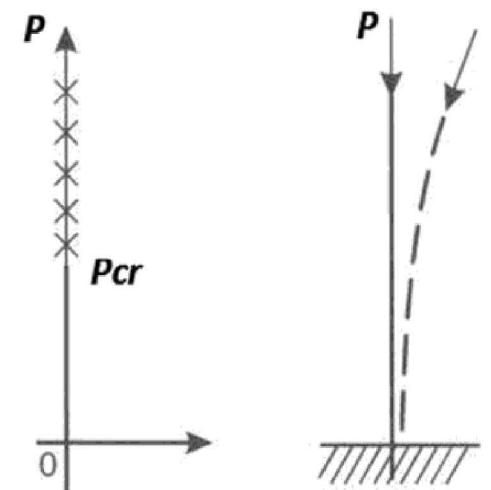


Figure 2.3. No equilibrium states

In addition to the described types, there are cases where critical states cannot exist despite greater-than-critical loads. In this case the consequence is not buckling as such but rather loss of the structure's load-bearing capacity.

Experience shows that even slight changes in critical load values can greatly stress the structure, causing buckling and failure. Therefore in operating conditions, the structure must never reach a critical state that precedes buckling. Thus new structure designs must be subjected to buckling analysis to determine critical loads that cause failure, as well as to evaluate the ability of the structure to maintain equilibrium.

Note that the loss of stable equilibrium can occur in elastic strain conditions, as well as when the material experiences stress greater than its elastic limit. A characteristic unstable state corresponds to every critical load value. The number of such states is equal to the number of degrees of freedom.

Engineering analysis tools implemented in **NX Advanced Simulation** with NX Nastran allow the solution of buckling problems using linear and nonlinear formulations.

2.1. Linear buckling analysis

In terms of FE modelling, the linear buckling problem is solved by adding a differential stiffness matrix to the linear stiffness matrix. From the physical standpoint, the differential stiffness characterizes a linear approximation of the effect of changing stiffness of an elastic system under increasing stress: the linear stiffness matrix is weakened with increased compression stress, and strengthened with increasing tension stress. The differential stiffness matrix is sometimes called the augmented stiffness matrix because it is added or subtracted from the linear stiffness matrix depending on loading conditions. In NX Nastran notation, the linear and differential stiffness matrices are denoted as $[K_a]$ and $[K_d]$ correspondingly [4].

The global stiffness matrix is of the following form:

$$[K] = [K_a] + [K_d] \quad (2.1)$$

Then the total potential energy of the system can be found using the following expression:

$$[U] = 0.5\{\boldsymbol{u}\}^T [K_a]\{\boldsymbol{u}\} + 0.5\{\boldsymbol{u}\}^T [K_d]\{\boldsymbol{u}\} \quad (2.2)$$

where $\{\boldsymbol{u}\}$ is the displacement vector of FE model nodes.

The energy method is used to determine critical values (bifurcation points on the curve of equilibrium states). This method involves tracking the change of total potential energy of the system as it transitions from one equilibrium state to another. A zero change of total potential energy corresponds to a bifurcation point [3]:

$$\frac{\delta[U]}{\delta u_i} = [K_a]\{\boldsymbol{u}\} + [K_d]\{\boldsymbol{u}\} = 0 \quad (2.3)$$

where u_i is the displacement of the i -th degree of freedom.

Therefore, the linear buckling problem can be solved by finding eigenvalues of the following characteristic equation:

$$([K_a] + \lambda[K_d]) = \{0\} \quad (2.4)$$

where λ is the eigenvalue corresponding to critical buckling load.

The number of positive roots λ of this equation is equal to the number of degrees of freedom of the FE model.

As a rule, only the first critical load value is significant, because in most cases, the structure fails before the next critical value is reached.

NX Nastran performs linear buckling analysis with the *SEBUCKL 105* solution. Eigenvalues can be found in linear buckling problems using one of the two methods [5]:

- Modified iterative method, using the Sturm sequence method. It is recommended for large problems if only a few lower eigenvalues need to be found.
- Lanczos method, which is used for problems of medium and large size. NX Advanced Simulation uses the Lanczos method by default.

Keep in mind that linear buckling analysis does not account for nonlinearities such as large displacements and strains, nonlinear material properties, and contact interaction. In some cases, these nonlinearities significantly influence the behaviour of the structure. The linear approach is efficient and convenient for determining critical loads at a preliminary stage before performing nonlinear buckling analysis. In NX Nastran, nonlinear analysis is performed using the *NLSTATIC 106* solution.

The following key limitations are present when solving linear buckling problems:

- Displacement vector components must be small.
- Stress and strain must increase linearly up to the buckling point.
- Post-critical behaviour of the structure is not considered.
- Differential stiffness is calculated for the following NX Nastran element types: *CONROD*, *CROD*, *CTUBE*, *CBAR*, *CBEAM*, *CBEND*, *CQUAD4*, *CQUAD8*, *CTRIA3*, *CTRIA6*, *CSHEAR*, *CHEXA*, *CPENTA*, and *CTETRA*.
- The distribution of internal forces in the element remains constant.
- Follower load effects are neglected.

Example 1. Linear buckling analysis

The following is a linear buckling analysis of a thin-shell part under compression. Shell elements are used in the FE model of the part.

1. Opening the CAD model and creating new model files

Launch NX and open the *strap.prt* file, then go to **NX Advanced Simulation: Start→Advanced Simulation**.

Reset all settings in the dialog boxes to default values. To do so, choose the main menu command: **Preferences→User Interface→General→Reset Dialog Box Settings**, click *OK*. Right-click the *strap.prt* file in the Simulation Navigator tab (Figure 2.4). Select **New FEM and Simulation** to create the FE model and the simulation model. In the FE model file creation dialog box, make sure the *Create Idealized Part* check box is selected, in the *Solver* list, select NX Nastran, then click *OK*. A new SIM file creation dialog box for the simulation model appears. In the **Solution** dialog box, select **Solution Type: SEBUCKLE 105**. Change the name of the new solution to *Linear_Buckling*.

Click **OK**.

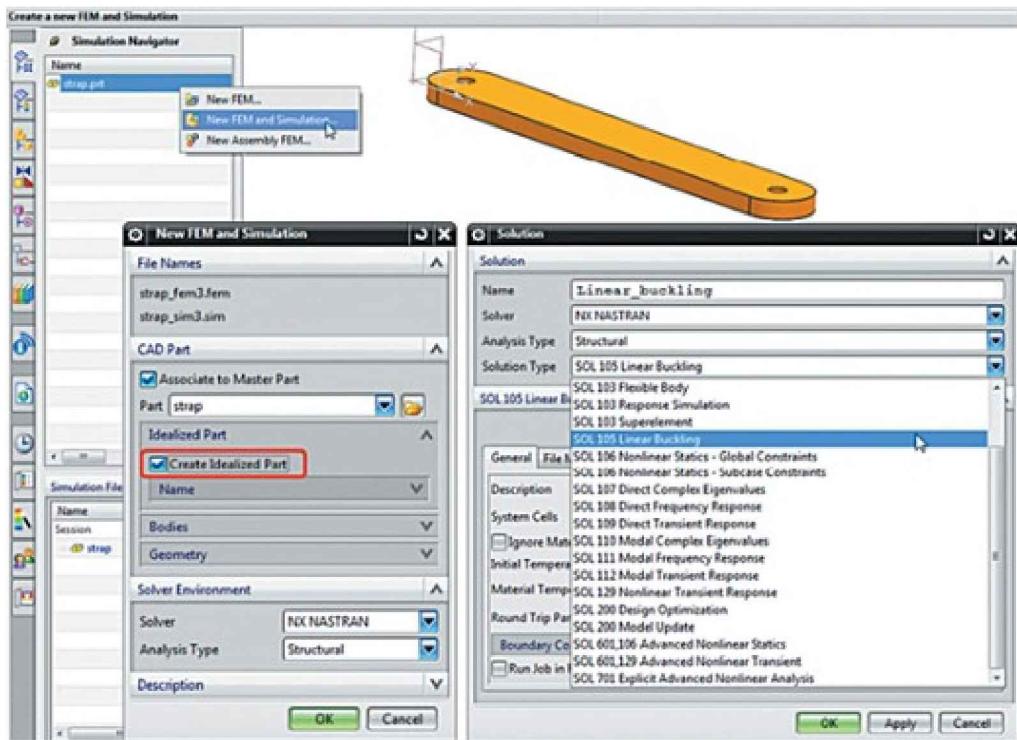


Figure 2.4. Creating new files of the FE model and the simulation model

Thus, based on the source geometry model, an idealized model (*strap_fem1_i.prt*), an FE model (*strap_fem1.fem*), and a simulation model (*strap_sim1.sim*) are created. The **Simulation File View** dialog box shows the names of these four files (Figure 2.5).

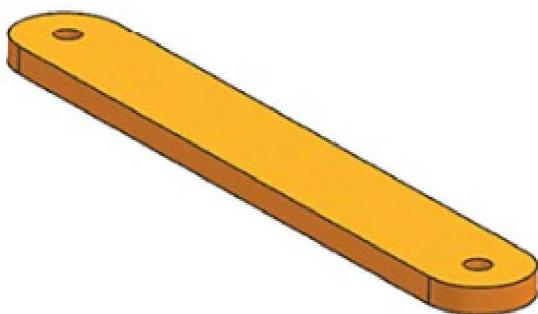


Figure 2.5. General view of the Simulation Navigator for the model

2. Creating a midsurface

2.1. Double-click the *Strap_fem1_i* file in the **Simulation File View** dialog box to switch to the idealized model. A message reminds you to create an associated copy of the source geometry model before performing any operations with the geometry. Click **OK**.

2.2. Use the **Promote** command on the body to create an associated copy. You can find this command on the **Advanced Simulation** toolbar.

2.3. Create a midsurface for the thin-shell body using the **Midsurface by face pairs** command on the **Midsurface** toolbar. In the dialog box, do the following (Figure 2.6):

- Select the part.
- Click *Automatically create face pairs*.
- Select the *Hide solid body upon apply* check box.
- Click **OK**.

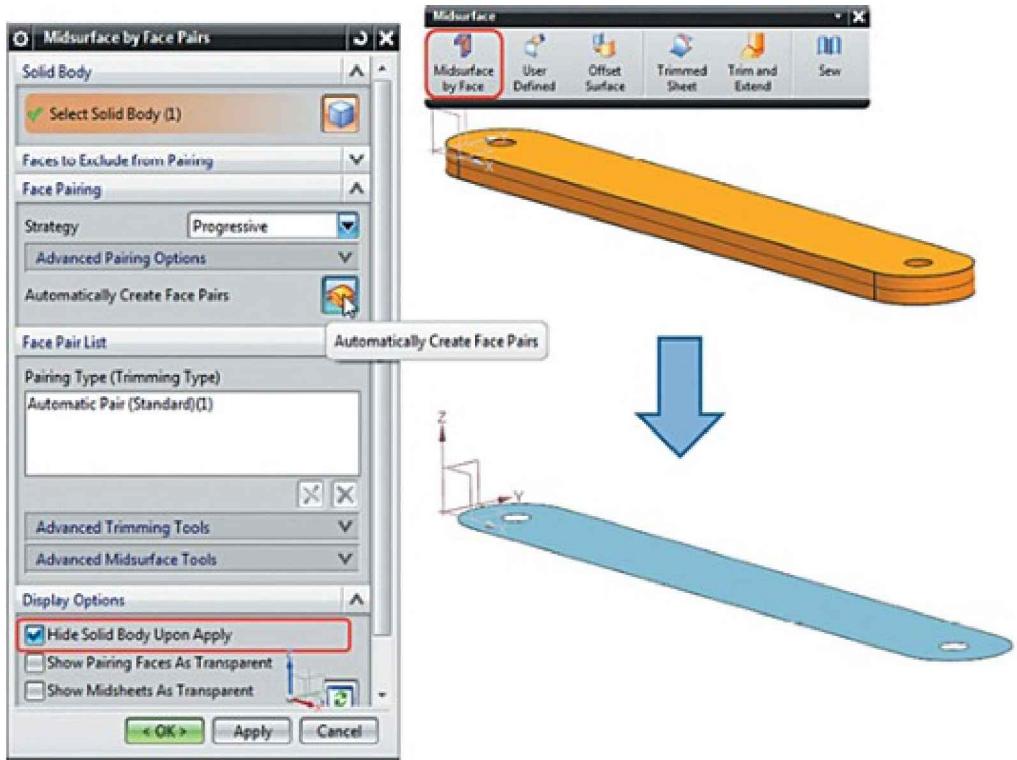


Figure 2.6. Creating a midsurface

3. Creating the FE model

- 3.1. Double-click the *Strap_fem1* file in the **Simulation File View** to switch to the FE model.
- 3.2. In the model tree, expand the *Polygon geometry* node and make the 3D *Polygon Body(1)* *unavailable*.
- 3.3. On the **Finite Element Model** toolbar, use the **Physical Properties** command. In the **Physical property tables manager** dialog box, specify the following settings (Figure 2.7):
 - In the *(Type* list, select *PSHELL*.
 - In the *Name* box, type *PSHELL1*.
 - Click **Create**.
 - In the new *PSHELL* dialog box, do the following (Figure 2.7):
 - Click **Choose material**. In the **Material List** dialog box, select *Steel material* and click **OK**.
 - In the *Default thickness* box, enter 4 mm.
 - Click **OK**.

- In the *(Type* list, select *PSHELL*.
- In the *Name* box, type *PSHELL1*.
- Click **Create**.
- In the new *PSHELL* dialog box, do the following (Figure 2.7):
 - Click **Choose material**. In the **Material List** dialog box, select *Steel material* and click **OK**.
 - In the *Default thickness* box, enter 4 mm.
 - Click **OK**.

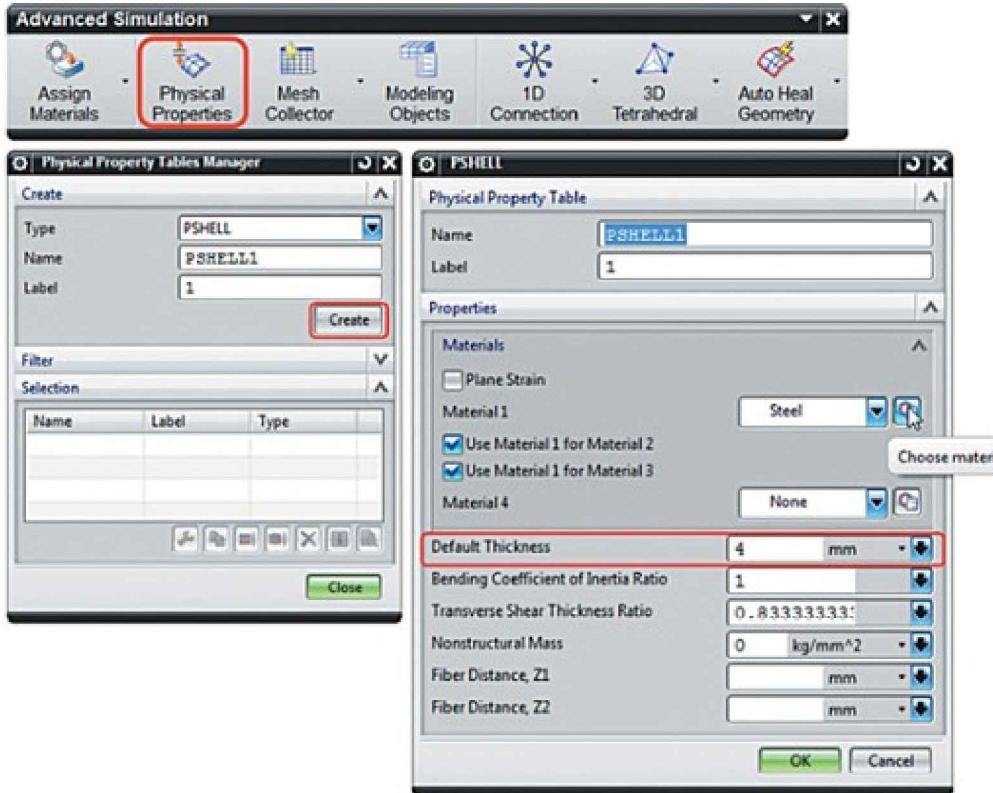


Figure 2.7. Creating physical properties of shell elements

3.4. The next step involves creating a mesh collector. Use the **Mesh collector** command of the NX Advanced Simulation toolbar and specify the following (Figure 2.8):

- In the *Element family* list, select *2D*.
- In the *Collector type* list, select *ThinShell*.
- In the *Type* list, select *PSHELL*.
- In the *Shell properties* list, select *PSHELL1*.
- In the *Name box*, type *Pshell1*.

Click **OK**.

Note that the model tree in the **Simulation Navigator** tab now contains a *2D Collectors* node with a *Pshell1* node (Figure 2.8).

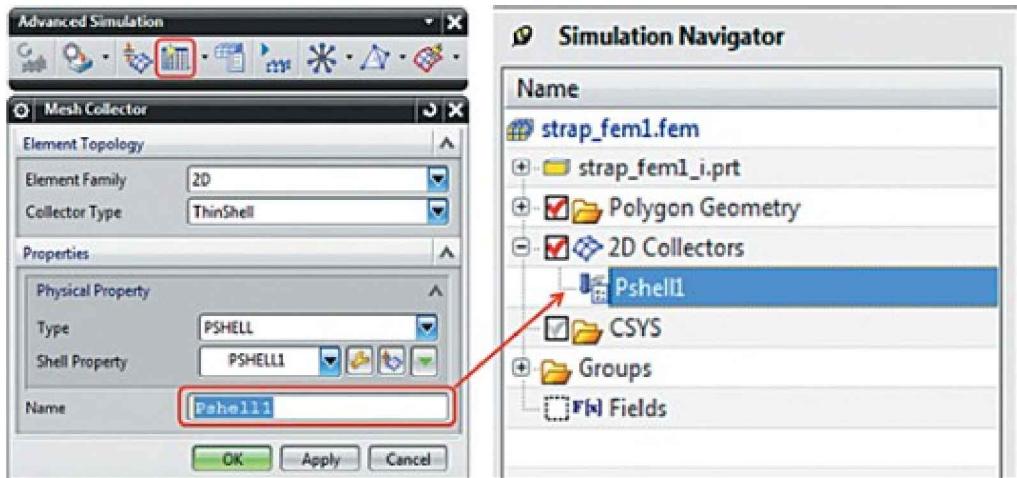


Figure 2.8. Creating a mesh collector

3.5. To generate an FE mesh, use the **2D Mesh** command on the Finite Element Model toolbar and specify the following (Figure 2.9):

- Click **Select objects** and select the midsurface of the part.
- In the **Type list**, select **CQUAD(4)**.
- In the **Meshing method** list, select **Paver**.
- In the **Element Size** box, enter 3 mm.
- Clear the **Automatic creation** check box and select **Pshell1** as the mesh collector.

Click **OK**.

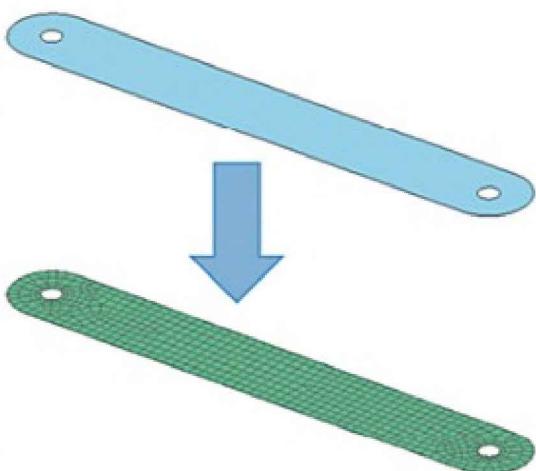


Figure 2.9. Creating an FE mesh

4. Setting up loads and boundary constraints

Double-click the **Strap_sim1** file in the **Simulation File View** to switch to the simulation model.

4.1. To create degree of freedom constraints for the structure, in the model tree of the Simulation Navigator tab, right-click **Constraint container** and choose **Fixed Constraint**, thereby constraining all degrees of freedom. In the dialog box, specify the following (Figure 2.10):

- Use **Select Object** to select an edge of one of the holes.

Click **OK**.

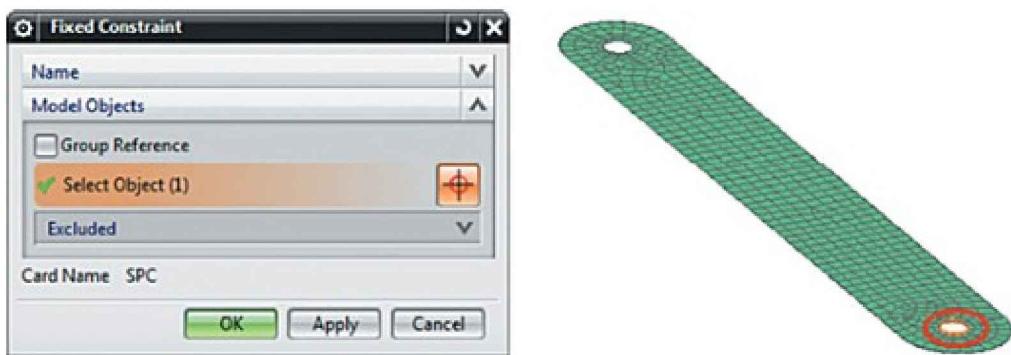


Figure 2.10. Constraining all degrees of freedom

4.2. In the model tree of the Simulation Navigator tab, right-click **Load container** to open the shortcut menu, hold the mouse pointer over **New Load**, and choose **Force** in the submenu. In the dialog box, specify the following (Figure 2.10):

- In the **Type** list, select **Magnitude and direction**.
- Click **Select Object** to select an edge of the unconstrained hole.
- In the **Force** box, enter **1 N**.
- Under **Specify vector**, specify the positive direction of OX axis.

Click **OK**.

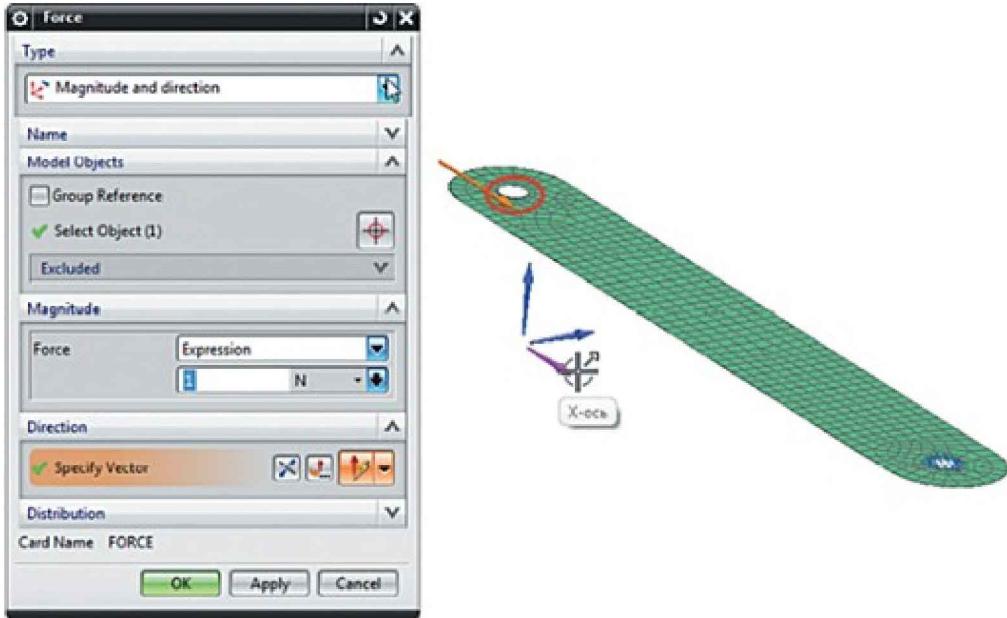


Figure 2.11. Specifying a compression force

5. Specifying solution parameters for the linear buckling problem

To set up solution parameters for the linear buckling problem, in the Simulation Navigator tab, right-click the *Subcase – Buckling Method* solution step, and choose *Edit* in the shortcut menu. In the **Solution Step** dialog box, specify the following (Figure 2.12):

- In the *Eigenvalue method* list, select *Lanczos*.
- Under *Lanczos Data*: Click **Create Modeling Object**. In the new **Real Eigenvalue – Lanczos1** dialog box, in the *Number of Desired Modes* box, type 10. Keep default values for the rest of parameters. Click **OK**.

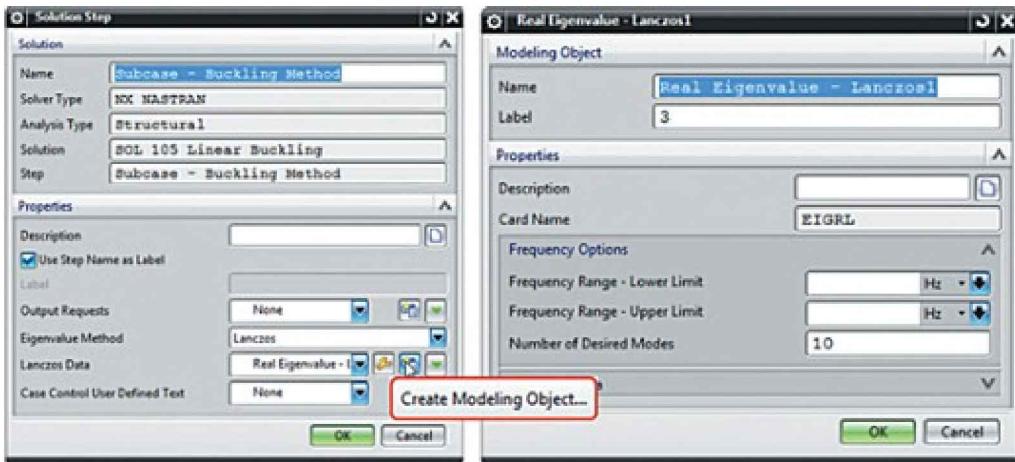


Figure 2.12. Solution parameters

The simulation model is ready for running. To save the model, choose **File→Save** in the main menu.

6. Running the solution

To run the solution of the linear buckling problem, right-click *Linear_Buckling* and choose **Solve...** in the main menu. In the ***Solution*** dialog box, click **OK**. When NX Nastran finishes, close all new dialog boxes.

7. Viewing results

In the model tree of the **Simulation Navigator** tab, double-click the **Results** node to switch to the **Post Processing Navigator** with results loaded. Note that the results are divided into two steps:

- *Subcase – Buckling Loads* contains static simulation results.
- *Subcase – Buckling Method* contains buckling simulation results, that is, eigenvalues and the corresponding buckling modes.

To view the results, expand the *Subcase – Buckling Method* node, expand *Mode 1*, and double-click *Displacement-Nodal*. The graphics area shows the deformed state that corresponds to the first buckling mode with an eigenvalue of $3.03E3$ Hz (Figure 2.13).

The critical load value is found by multiplying the applied load value and the determined eigenvalue. In this example, the applied load value is 1 N, therefore, critical load value for the first buckling mode is $3.03E3$ N.

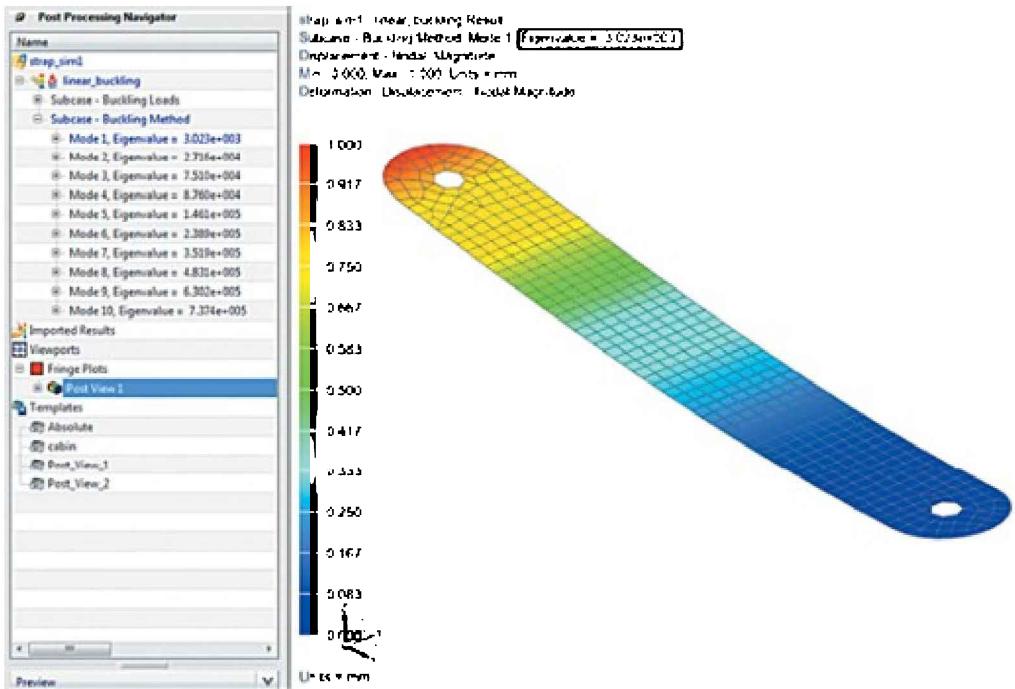


Figure 2.13. Viewing simulation results

To close the simulation model, choose **File→Close→All Parts** in the main menu.

2.2. Nonlinear buckling analysis

Linear buckling analysis often overestimates critical load values because the solution does not account for various nonlinearities of the structure. To obtain more realistic results and analyse the post-critical behaviour of the structure, nonlinear buckling analysis with *NLSTATIC 106* solution type is necessary. Nonlinear buckling problems can account for the following nonlinearity types:

- Elastoplastic material properties.
- Geometrical nonlinearity that accounts for large displacements and the change of force direction with changing deformed state of the structure (follower load effect).

NX Nastran uses two converged solutions in immediate vicinity of the buckling point to create the differential stiffness matrix.

There are two approaches to finding buckling points in the narrow nonlinear

region [6]:

- Tangential stiffness matrix $[K]$ (2.1) is proportional to the external load, meaning that critical load P_{cr} can be linearly interpolated:

$$P_{cr} = P_n + \alpha \Delta P, \quad \Delta P = P_n - P_{n-1} \quad (2.5)$$

where α is the *critical buckling factor*.

- Tangential stiffness matrix $[K]$ is proportional to the increment of displacement, therefore displacement U_{cr} can be found by extrapolating from the current state:

$$U_{cr} = U_n + \lambda \Delta U, \quad \Delta U = U_n - U_{n-1} \quad (2.6)$$

NX Nastran uses the second approach. It is assumed that the tangential stiffness matrix changes linearly, therefore internal forces are a quadratic function of displacement. Nonlinear buckling analysis is activated by the *BUCKLE* parameter [8].

If a negative definite matrix is found in nonlinear buckling analysis, it should be restarted from the last step with a positive definite matrix.

Before running a nonlinear buckling analysis, it is recommended to do the following:

- Perform a linear buckling analysis to find the critical load without consideration of nonlinear behavior of the structure.
- Perform a nonlinear static analysis with *NLSTATIC 106* solution to find the nearest point to the buckling point. This simulation should be performed with small increments until negative components appear in the stiffness matrix.

When solving nonlinear buckling problems, make sure the stiffness matrix is updated at each iteration, that is, enter a value of 1 for the *Iterations before update* parameter (Figure 2.19).

Example 2. Nonlinear buckling analysis

Launch NX and open the *strap_fem1.fem* file. You will automatically switch to **NX Advanced Simulation**.

Reset all settings in the dialog boxes to default values. To do so, choose the main menu command: Preferences→User Interface→General→Reset Dialog Box Settings, click **OK**.

1. Create a new simulation model file

Right-click *strap_fem1* in the Simulation Navigator tab. Choose **New Simulation** and enter *strap_sim2* as the name. Click **OK**. In the new dialog box, click **OK**. In the **Solution** dialog box, do the following:

- In the *Name* box, type *NlN_Static*;
- In the *Solution Type list*, select *NLSTATIC 106*.
- In the *Case Control* tab, make sure that for the *Nonlinear Parameters* option the default *Nonlinear Parameters1* modelling object is selected. Click *Edit*. In the **Nonlinear Parameters1** dialog box (Figure 2.14) specify the following:
 - In the *Number of Increments* box, type 20.
 - In the *Intermediate Output Flag*, select *All* to write results at each iteration.

Keep default values of the other parameters, click **OK**.

- In the *Parameters* tab, select the *Large Displacements* check box.

Click **OK** to close the **Solution** dialog box.



Figure 2.14. Specifying nonlinear solution parameters

2. Specifying constraints on degrees of freedom

Proceed in accordance with para. 4.1 of Example 1 (Figure 2.10).

3. Specifying loads

Specify a 3400 N load for the nonlinear static analysis, slightly greater than the critical load value of 3030 N obtained by solving the linear buckling problem for this part (Example 1).

If nonlinear static analysis is performed under longitudinal compressive load only, the part experiences only compressive strain and does not buckle. This is because the FE model does not account for the heterogeneity of material structure, geometry imperfections, manufacturing defects and the like. Therefore, in addition to the compressing force, specify a slight transverse load of 1 N to create an additional perturbation.

3.1. Specify the compression force along the OX axis in accordance with para. 4.2 of Example 1, but instead of 1 N specify 3400 N.

3.2. Specifying a slight transverse load

In the model tree of the Simulation Navigator tab, right-click *Load container* to open the shortcut menu, hold the mouse pointer over **New Loads**, and choose *Force* in the submenu. In the dialog box, specify the following:

- In the *Type* list, select *Magnitude and direction*.
- Click *Select Object* to select nodes in the central region of the part as shown in Figure 2.15.
- In the *Force* box, enter *1 N*.
- Click *Specify vector* and specify the positive direction of the OZ axis.

Click **OK**.

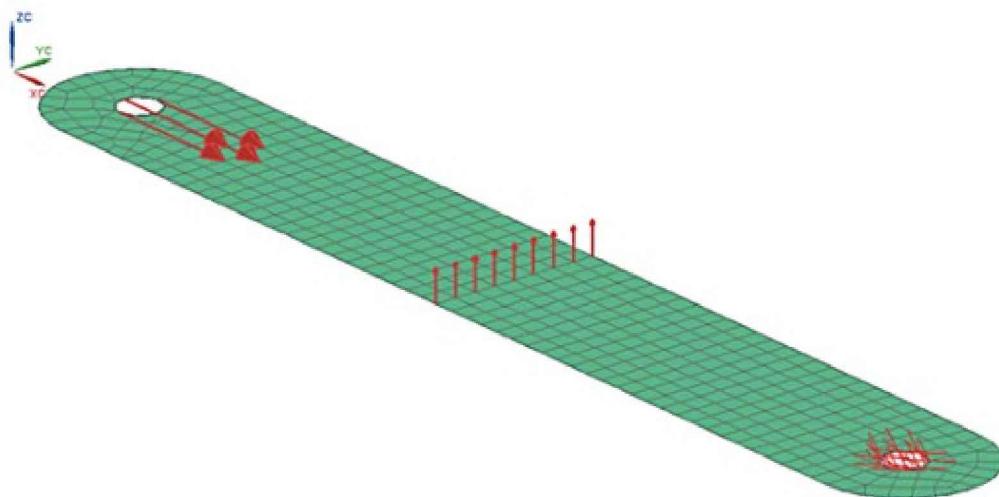


Figure 2.15. Loads and boundary conditions

4. Specifying nonlinear material properties

In the model tree of the **Simulation Navigator** tab, expand the *strap_fem1.fem node*, right-click *Pshell1* and choose *Edit Attributes Overrides*. In the dialog box, click *Type* and select *Apply override* (Figure 2.16). Click *Open Manager*. In the *Physical Property Tables Manager* dialog box, in the *Type* list, select *PSHELL* and click *Create*.

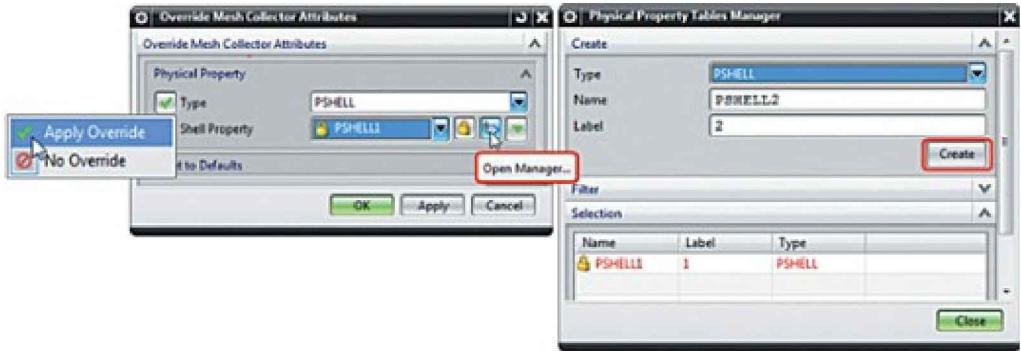


Figure 2.16. Creating physical properties

In the *PSHELL* dialog box, set up the following options:

- In the *Default Thickness* box, enter *4 mm*.
- Under *Material 1*, click *Choose material*. In the *Material List* dialog box, select *Steel* material and click *Copy the selected material* (Figure 2.17).

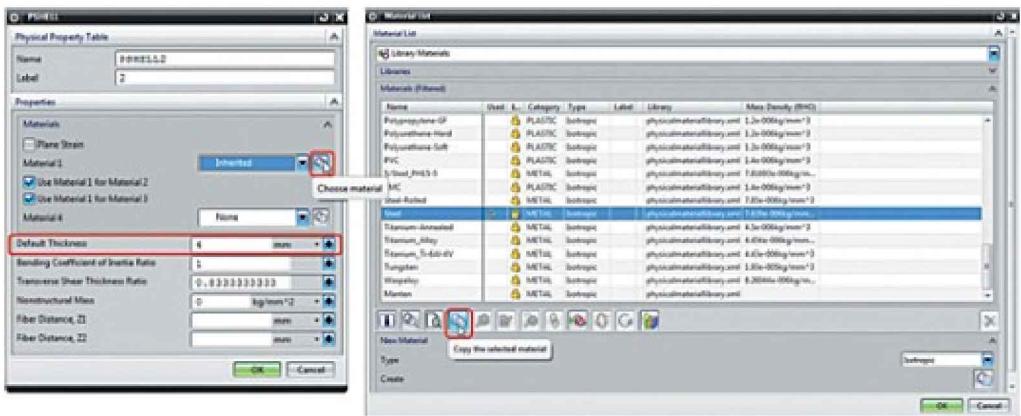


Figure 2.17. Specifying physical properties

In the *Isotropic material* dialog box, in the *Mechanical* tab, in the *Stress-Strain Related Properties* group, specify the following (Figure 2.18):

- In the *Initial Yield Point box*, enter *269.1 MPa*.
- In the *Stress-Strain list*, select *Field*.
- Under *Define Field*, click *Table Constructor*. In the *Table Field* dialog box, specify tabular data for the material strain curve (Figure 2.18):
 - a. In the *Domain* group, in the *Independent* tab, select *1-D General* in the drop-down list.

- b. In the *Data Points* group, click *Import from file* and specify the *Strain_stress.csv* file, which contains the necessary tabular data.

Click **OK** in all dialog boxes.

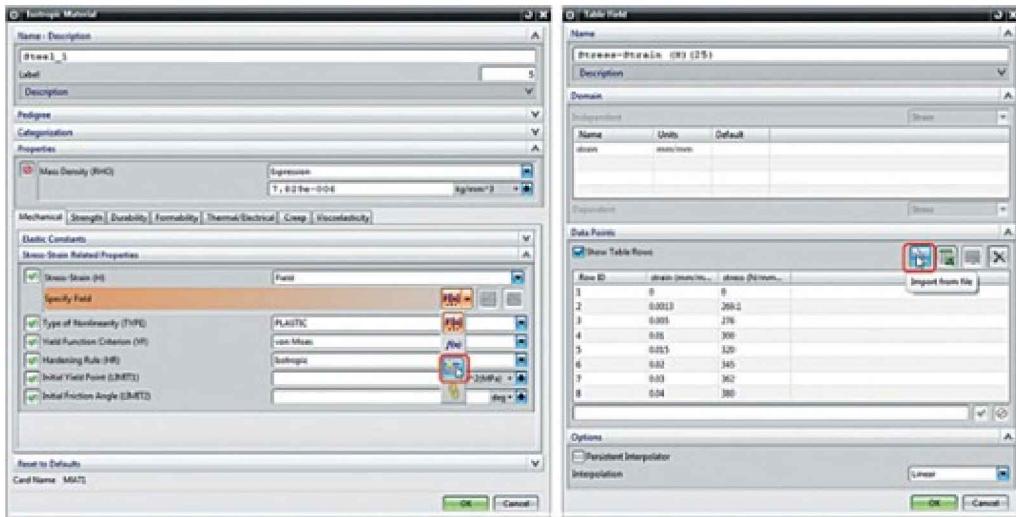


Figure 2.18. Specifying nonlinear material properties

Note that the *Pshell1* node in the model tree of the **Simulation Navigator** tab is highlighted in red. This means that physical properties of finite elements in the simulation model are overridden. Remember that the physical properties specified for the FE model remain unchanged.

The simulation model is ready for solving the nonlinear buckling problem. To save the model, choose **File→Save** in the main menu.

5. Running the solution

To run the solution, right-click *N/n_Static* and choose **Solve...** in the shortcut menu. In the new dialog box, click **OK**. The computation can take several minutes. When NX Nastran finishes running, close all new dialog boxes.

6. Viewing simulation results

In the model tree of the **Simulation Navigator** tab, double-click *Results* to select the **Post Processing Navigator** tab, and then expand the *N/n_Static* node. Note that the solution stopped running at the twentieth loading step but full load was not achieved (the load factor was 0.8797). Since nonlinear static analysis is performed with an increment of 0.05, buckling must occur

between load factor values 0.85 and 0.9, that is, in the load range of 2890 N through 3060 N.

7. Creating a new nonlinear solution for buckling analysis

In the model tree of the Simulation Navigator tab, right-click *Strap_sim2.sim*, and choose *New Solution*. In the *Solution* dialog box, specify the following:

- In the *Name* box, type *Nln_Buckling*.
- In the *Solution Type* list, select *NLSTATIC 106*.
- In the *Case Control* tab, under *Nonlinear Parameters*, click **Create Modeling Object**. In the **Nonlinear Parameters2** dialog box, specify the following (Figure 2.19):
 - In the *Number of Increments* box, type 20.
 - In the *Intermediate Output Flag* list, select *All* to save results at each iteration.
 - In the *Iterations Before Update* box, type 1 to update the stiffness matrix at each iteration.

Keep default values of the other parameters, and click **OK**.

In the *Parameters* tab of the **Nonlinear Parameters2** dialog box, select the *Large Displacements* check box and click **OK**.



Figure 2.19. Specifying nonlinear solution parameters

8. Specifying Subcase parameters for nonlinear buckling analysis

In the model tree of the Simulation Navigator tab, right-click the *Subcase – NL Static Loads1* node and choose *Edit* in the shortcut menu. In the *Solution Step* dialog box, set the following (Figure 2.20):

- In the *Analysis type* list, select *Buckling*.
- In the *Eigenvalue method* list, select *Lanczos*.
- In the *Lanczos Data* drop-down list, select the *Real Eigenvalue – Lanczos1* modelling object.

Click **OK**.

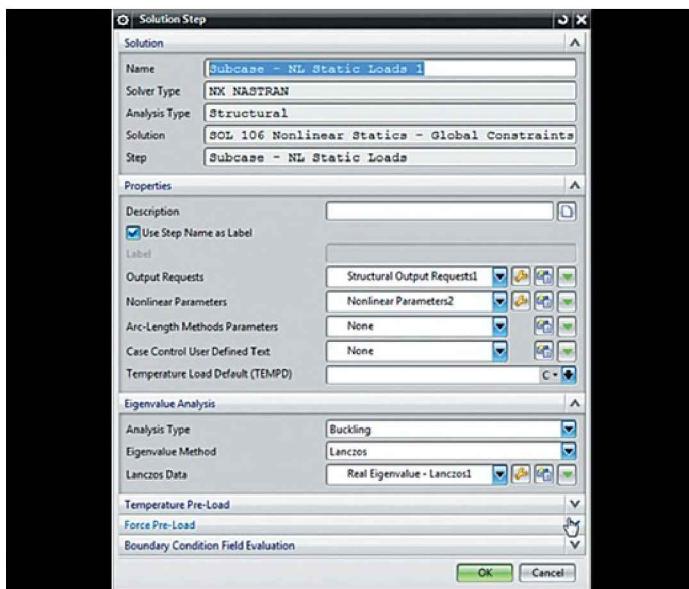


Figure 2.20. Specifying simulation case parameters

9. Specifying loads and displacement constraints

- 9.1. In the model tree of the Simulation Navigator tab, expand the *Constraint container* node and drag the *Fixed(1)* constraint into the *Constraints* node of the *NIN_Buckling* solution (Figure 2.21). In a similar way, drag the *Force (2)* load into the *Loads* node of the simulation case *Subcase – NL Static Loads1*.

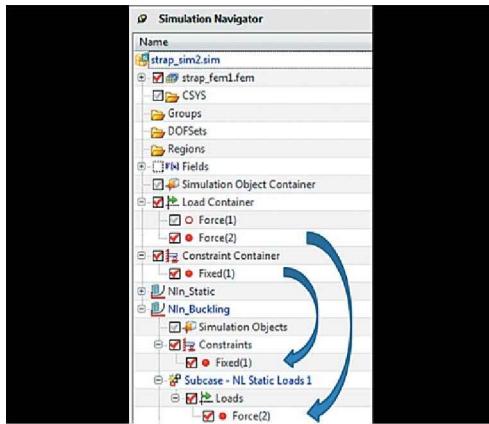


Figure 2.21. Specifying loads and constraints

9.2. In the model tree of the Simulation Navigator tab, right-click the *Loads* node of the *Subcase – NL Static Loads1* simulation case, hold the mouse pointer over **New Loads**, and choose **Force** in the submenu. In the dialog box, specify the following:

- In the *Type* list, select *Magnitude and direction*.
- Click *Select Object* and select the edge of the hole that has no pinned constraints defined.
- In the *Force* box, enter *2800 N*.
- Under *Specify vector*, specify the positive direction of OX axis.

Click **OK**.

The simulation model is ready for running. To save the model, choose **File→Save** in the main menu.

10. To run the solution, right-click *Nln_Buckling* and choose **Solve...** in the shortcut menu. In the **Solution** dialog box, click **OK**. The computation can take several minutes. When NX Nastran finishes running, close all new dialog boxes.

11. Viewing simulation results

Find the *strap_sim2-nln_buckling file.f06* in the working folder and open it in a text editor. Find the string: CRITICAL BUCKLING FACTOR (ALPHA) = 1.212492E+00. In accordance with formula (2.5), the critical load for the calculated critical buckling factor is:

$$P_{cr} = P_n + \alpha \Delta P = 2800 + 1.2 * 2800/20 = 2968 \text{ N.}$$

Chapter 3. Dynamic analysis

Most structures experience operating loads that vary with time, therefore, designing a new state-of-the-art product that satisfies reliability and safety criteria requires dynamic (time based) analysis in addition to static analysis. Dynamic analysis involves analysing the response of the structure to external influences that vary with time. Dynamic analysis finds the structure's response, which manifests as displacements, velocities, accelerations, reaction forces, strain, and stress as functions of time.

Studies of dynamic behaviour of structures seek to achieve the following basic objectives:

- Evaluation of vibration resistance, stability, and endurance of structures.
- Using dynamic analysis results as a history of loading to analyse fatigue strength of a structure subjected to cyclical loads.
- Enhancing the design and/or evaluating the possibility of using additional structural elements to reduce vibration and eliminate resonance in operation.
- Development of vibration protection measures to reduce the adverse influence of vibration on operating personnel who directly service the equipment.

Modern techniques allow replacing field trials, which require significant allocations of time and money, with numerical modelling, using state-of-the-art software for a broad spectrum of analyses. Availability and constant improvement of computing systems based on FE modelling, allows the solution of many difficult dynamic problems without resorting to experimental methods.

This chapter describes the basic features implemented in the **Advanced Simulation** application that use NX Nastran to carry out dynamic analysis of structures.

In NX Nastran, the following three basic types of dynamic problems can be distinguished:

- Finding natural frequencies and free vibration modes without damping (real or normal modes of vibration). The *SEMODES 103* solution is used.
- Analysis of dynamic behaviour of the structure under short-term loading (unsteady-state oscillatory process), in other terms, analysis of transient processes in the structure. Depending on the approach, this analysis type can use either direct (*SEDTRAN 109 – Direct Transient Response*) or modal (*SEMTRAN 112 – Modal Transient Response*) methods of solving the equation of motion.
- Frequency analysis is the analysis of a structure's response to a steady-state, relatively long-term harmonic excitation. You can use direct (*SEDFREQ 108 – Direct Frequency Response*) and modal (*SEMFREQ 111 – Modal Frequency*) methods for this analysis as well.

In addition, dynamic analysis allows you to determine the response of structures to dynamic effects such as impacts or random vibration. You can also solve hydroelasticity and aeroelasticity problems or use superelements and dynamic reduction in calculations. Note that all these dynamic analysis method types have their own peculiarities, and may be interdependent. For example, natural frequencies and modes are often found at a preliminary stage of other types of dynamic analyses.

In addition to basic types, there are niche NX Nastran solutions for some classes of problems:

- *SEMODES 103 – Flexible Body* is the solution for calculating natural frequencies and modes of a mechanical part to carry out dynamic reduction. It is used for kinematic analysis of a mechanism in **NX Motion Flexible Body**, factoring in the flexibility of its structural elements (kinematic links).
- *SEMODES 103 – Superelement* is the solution for calculating natural frequencies and modes of a component of the general simulation model, followed by dynamic reduction. This solution is useful when external superelements (subsystems) of a structure are created.
- *SEMODES 103 – Response Simulation* is the solution for calculating natural frequencies and modes of a structure for use in

NX Response Simulation.

Some classes of problems, for example, rotor dynamics, require complex natural frequencies to be found. With the direct method, you can use the *SEDCEIG 107 – Direct Complex Eigenvalues*, solution, and with the modal method, you can use the *SEMCEIG 110 – Modal Complex Eigenvalues* solution.

You can use the **NX Response Simulation** module built into the **NX Advanced Simulation** pre-post processor to quickly analyse dynamic behaviour of structures using the normal mode superposition method. It can be used to carry out unsteady-state analysis, frequency analysis, random vibration analysis, spectral analysis, and DDAM (Dynamic Design Analysis Method) analysis.

3.1. Fundamentals of dynamic analysis of structures

If the system moves under the influence of external forces that vary with time, the arising internal forces are functions of time. The motion of a system is characterized by four fundamental properties: mass, energy dissipation, stiffness of the system, and external loads. The equation that characterizes the equilibrium state of the system at each moment of time is called the equation of motion. For a system with one degree of freedom, this equation is of the form:

$$m\ddot{u}(t) + b\dot{u}(t) + ku(t) = p(t) \quad (3.1)$$

where m is the mass.

b is the damping coefficient.

k is the stiffness factor.

$u(t)$ is the displacement.

$p(t)$ is the external force.

The left-hand part of this equation corresponds to internal forces that arise in the dynamic system:

$m\ddot{u}(t)$ is the force of inertia.

$b\dot{u}(t)$ is the viscous damping force. It is a function of the damping coefficient and velocity. It characterizes the process of energy dissipation due to conversion of kinetic energy to heat.

$ku(t)$ is the elastic resistance force, a function of stiffness and displacement of the system.

The right-hand part of the equation contains the external load $p(t)$ that varies with time.

There are two types of dynamic analysis: analysis of free vibration and driven vibration.

Analysis of free vibration without accounting for dissipative properties of the system involves solving an equation of motion of the following form:

$$m\ddot{u}(t) + ku(t) = 0 \quad (3.2)$$

The solution of this equation is the following equation:

$$u(t) = A\sin\omega_n t + B\cos\omega_n t \quad (3.3)$$

where $u(t)$ is the sought displacement as a function of time.

$A = \dot{u}_0/\omega_n$, $B = u_0$ are constants determined from the initial conditions of the system.

$\omega_n = 2\pi f_n$ is the natural angular frequency.

f_n is the natural cyclic frequency.

The number of natural frequencies of a system is equal to its number of degrees of freedom.

If dissipative forces are taken into account, the equation of motion (3.2) is of the form:

$$m\ddot{u}(t) + b\dot{u}(t) + ku(t) = 0 \quad (3.4)$$

In terms of dynamic analysis there is a concept of critical viscous damping coefficient that ensures the absence of vibration in a system if its resting state is disturbed. The critical damping coefficient is defined by the expression:

$$b_{cr} = 2\sqrt{km} = 2m\omega_n \quad (3.5)$$

Dynamic behavior of a system significantly depends on the ratio of viscous damping coefficient to the critical viscous damping coefficient.

If the damping coefficient is greater than or equal to the critical damping coefficient, the solution of equation (3.4) is of the form:

$$u(t) = (A + Bt)e^{-bt/2m} \quad (3.6)$$

In this case the system returns to the initial state exponentially without periodic oscillations (Figure 3.1).

If the damping coefficient is less than the critical value (the most typical case), then solution of equation (3.4) is of the form:

$$u(t) = e^{-bt/2m}(Asin\omega_d t + Bcos\omega_d t) \quad (3.7)$$

where ω_d is the natural angular frequency with damping taken into account.

The ratio of natural angular frequency with damping ω_d to natural angular frequency of the system without damping ω_n is found using the following expression:

$$\frac{\omega_d}{\omega_n} = \sqrt{1 - \zeta^2} \quad (3.8)$$

where ζ is the damping (aperiodicity coefficient), which characterizes the amount of viscous damping in the system in fractions of critical damping:

$$\zeta = \frac{b}{b_{cr}} \quad (3.9)$$

In this case the system returns to the resting state through exponential abatement of the periodic oscillation amplitude from cycle to cycle (Figure 3.1).

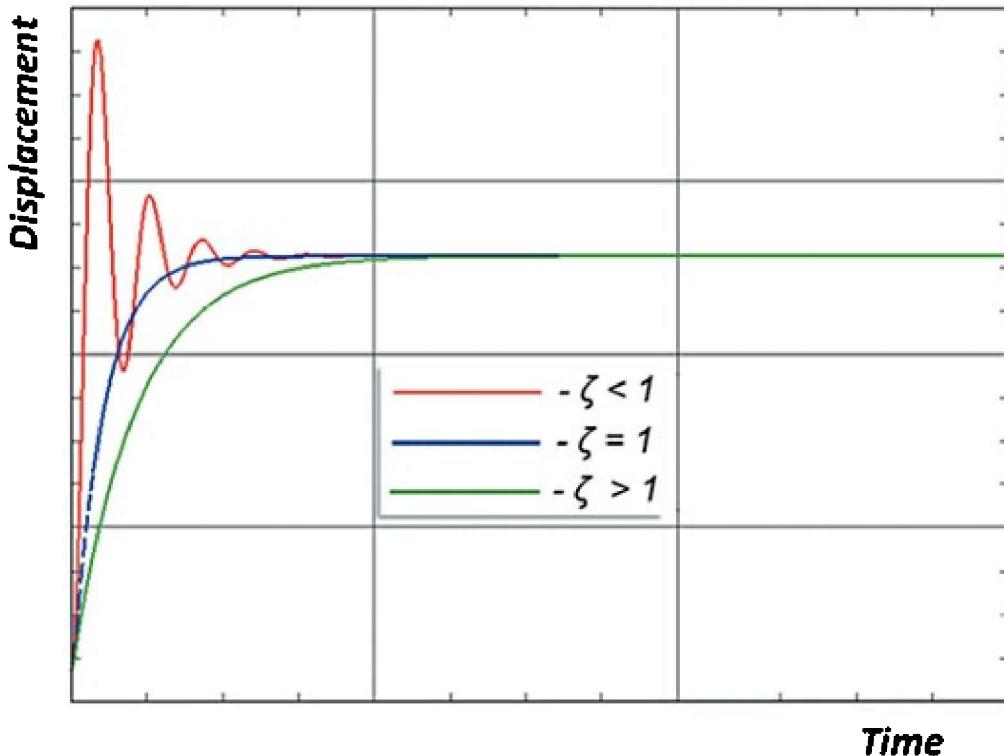


Figure 3.1. Free vibration of the system with different values of ζ

If external harmonic action is present, it constitutes a driven vibration problem, and the equation of motion for one degree of freedom without damping is of the form:

$$m\ddot{u}(t) + ku(t) = p(t) \quad (3.10)$$

where $p(t) = \sin\omega t$ is the driving force.

p_0 is the amplitude of the driving force.

ω is the frequency of the driving force.

The solution of equation (3.10) with driving input frequency ω is:

$$u(t) = A\sin\omega_n t + B\cos\omega_n t + \frac{p_0/k}{1 - \omega^2/\omega_n^2} \sin\omega t \quad (3.11)$$

where A and B are constant values determined from initial conditions.

p_0/k is the static offset due to the amplitude value of the driving force.

Let us introduce the so-called dynamic amplification factor Q :

$$Q = (1 - \omega^2/\omega_n^2)^{-1} \quad (3.12)$$

If the frequency of the external input ω is significantly lower than the natural frequency ω_n of the system, the oscillation amplitude corresponds to the static offset. As the frequency of the external force ω increases, the amplitudes of oscillations increase as well. If the frequency ω is equal to the natural frequency of the system ω_n , the amplitude of oscillations sharply increases and the offset $u(t)$ tends to infinity. This phenomenon is called resonance.

If damping forces are taken into account, the equation of motion for driven oscillations is of the form:

$$m\ddot{u}(t) + b\dot{u}(t) + ku(t) = p_0 \sin \omega t \quad (3.13)$$

The solution of this equation for the case with steady-state driven oscillations is:

$$u(t) = p/k \frac{\sin(\omega t + \theta)}{\sqrt{(1 - \omega^2/\omega_n^2)^2 + (2\zeta\omega/\omega_n)^2}} \quad (3.14)$$

where θ is the phase angle or phase shift that characterizes the advance of the response to an external influence (phase lead) as defined by the expression:

$$\theta = -\tan^{-1} \frac{2\zeta\omega/\omega_n}{(1 - \omega^2/\omega_n^2)} \quad (3.15)$$

In this case the dynamic amplification factor Q with damping is of the form:

$$Q = \left(\sqrt{(1 - \omega^2/\omega_n^2)^2 + (2\zeta\omega/\omega_n)^2} \right)^{-1} \quad (3.16)$$

The relative magnitudes of external input frequency ω and the natural frequency ω_n of the system determines the system's behaviour:

- If $\omega \ll \omega_n$, the dynamic amplification factor Q tends to one, the amplitude of oscillations corresponds to static offset p_0/k in phase with the load.
- If $\omega \gg \omega_n$, the dynamic amplification factor tends to zero, and the

phase angle is 180° . The load changes very rapidly, so the system cannot react to the external influence fast enough.

- If $\omega = \omega_n$, a resonance condition exists. Unlike driven vibration without damping, the dynamic ratio is not infinite and is equal to $0.5\zeta^{-1}$. The response of the system leads the load by 270° .

3.2. Accounting for inertial and elastic damping properties

When using the finite-element method to model dynamic behaviour of systems, the equation of motion with several degrees of freedom has the following form in matrix notation:

$$[M]\{\ddot{u}\} + [B]\{\dot{u}\} + [K]\{u\} = \{P(t) \text{ или } P(\omega)\} \quad (3.17)$$

where $[M]$ is the mass matrix, $[B]$ is the viscous damping matrix, $[K]$ is the stiffness matrix, $\{P(t) \text{ or } P(\omega)\}$ is the external influence as a function of time (transient analysis) or a function of frequency (frequency analysis), $\{\ddot{u}\}$, $\{\dot{u}\}$, $\{u\}$ are vectors of nodal accelerations, velocities, and displacements of the FE model.

Equation (3.17) can be solved to obtain displacements, accelerations, velocities, stresses, strains, and forces at nodes of the FE model as functions of time.

Finite-element dynamic analysis has associated peculiarities, prerequisites, and assumptions that apply to the simulation model. These are necessary to correctly describe dynamic properties of the system being analysed. This is mostly relevant for specifying mass properties and damping coefficients. It is notable that in dynamic analysis, you can determine the response without directly modelling special vibration damping devices (shock absorbers, dampers, compensators and so on). These structural elements are described using NX Nastran spring-damper finite elements (CVISC, CDAMPi, CBUSH, CELASi) with correct stiffness and damping properties.

3.2.1. Mass properties

The mass matrix in NX Nastran can be defined in several ways:

- Specifying the density of the material, which allows you to determine the mass of finite elements directly. Density can be

defined by entering the relevant value in the new material creation dialog box, by selecting an existing material from the library, or in the *RHO* field of the *MAT1* card [8].

- If stiffness properties of some additional elements can be neglected but their weight is substantial (for example, the weight of snow cover or thermal insulation), the so-called non-structural mass is used. This type is defined for 1D finite elements as a value characterizing the ratio of mass to length, and for 2D finite elements as the ratio of mass to area. Nonstructural mass is specified in the corresponding dialog box when you set up physical properties of finite elements (for example, *PSHELL* for shell 2D elements, *PBEAM* for 1D elements) (Figure 3.2), or in the **NSM** (non-structural mass) field of *PSHELL* or *PBEAM* cards [7, 8].

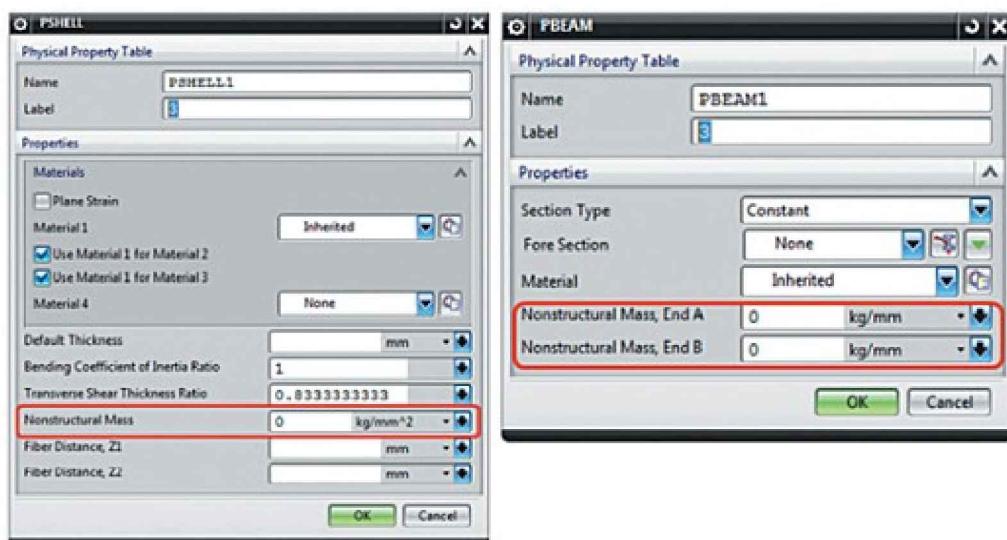


Figure 3.2. Specifying non-structural mass

- Specifying mass properties using concentrated masses. You can achieve this with *CONMi* or *CMASSi* 0D finite elements. Mass properties are specified for elements of the *CONMi* type in the **Mesh Associated data** dialog box, for *CMASSi* – *PMASS* type elements they are specified in element properties (Figure 3.3) or in *CONMi* or *PMASS* cards respectively [7, 8].

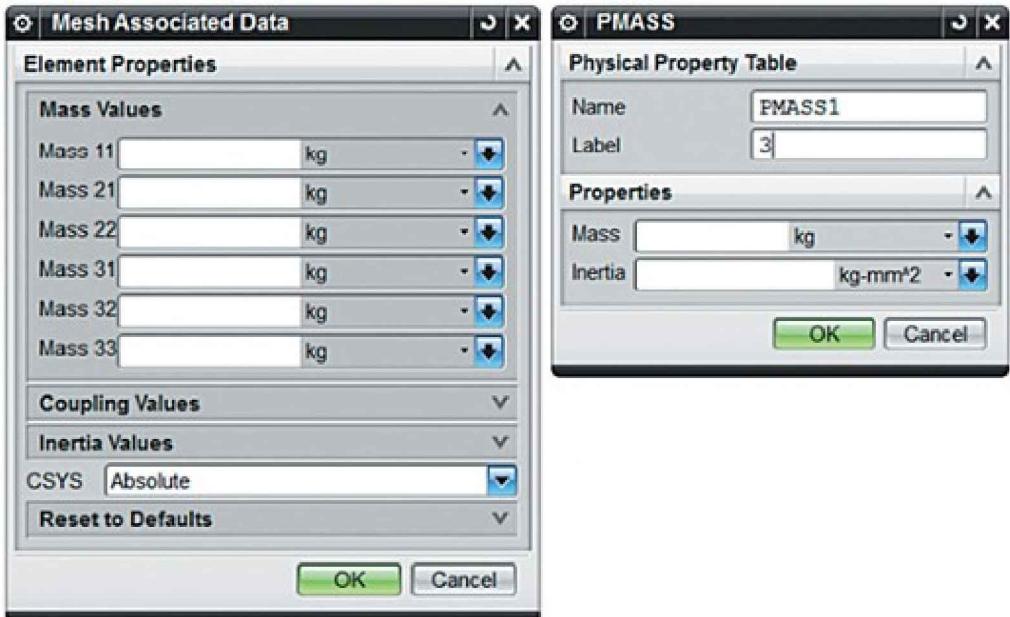


Figure 3.3. Specifying concentrated masses

3.2.2. Damping

Damping is introduced to describe energy dissipation in a structure that moves under external dynamic influence. Exactly describing the energy dissipation mechanism by introducing damping coefficients is often quite difficult because many design factors can influence the general behaviour of a dynamic system:

- Damping capacity of the material (internal friction). In cyclic damping, the material of the body irreversibly absorbs a part of the work of external forces, which then dissipates as heat.
- External friction is the loss of energy in points of attachment and connections between structural elements (flanges, bolted connections, joints, and so on).
- Damping mechanisms, that is, devices for damping vibration (dampers, vibration compensators, hydraulic and pneumatic shock absorbers, and so on).

Analytically finding each component's contribution to the total damping of the system is a complicated task. Therefore, field trials are often used to find precise damping values. However, experience shows that approximate damping values are sufficient for dynamic problems because damping is

typically insignificant and is 0.02 to 0.1 of the critical value for most structures. In some situations the damping of the system can be neglected altogether. For example, in short-term loading (impact), damping may be neglected because the maximum response of the structure occurs before the energy dissipation process begins. Expression (3.8) shows that if you find natural frequencies of free vibration for a system with a damping coefficient of 0.1, the eigenvalues scarcely differ from the actual values of a system without damping.

Introducing damping is important if there is long-term, steady-state influence on the system, and to obtain a correct response of the structure in resonance conditions.

NX Nastran uses the following parameters to describe damping properties of systems [7]:

- Damping coefficient or aperiodicity coefficient ζ .
- Dynamic amplification factor or the quality of the system Q .
- Viscous damping coefficient b .
- Structural (hysteretic) damping coefficient G .

The following is a brief description of these parameters.

The aperiodicity coefficient or damping coefficient (ζ) is a dimensionless value that characterizes the degree of energy dissipation due to internal friction in the material of a dynamic system experiencing cyclic deformation of its structural elements. In other words, this coefficient characterizes the damping in the system as a fraction of the critical damping. It can be found using the following expression:

$$\zeta = \frac{b}{b_{cr}} \quad (3.18)$$

where b_{cr} is the critical damping coefficient.

ω_n is the natural angular frequency of the system.

The dynamic amplification factor (the Q-factor) is inversely proportional to the energy released in a single oscillation cycle. If the natural frequency and the external influence frequency are equal, the Q-factor of the system is determined by the expression:

$$Q = \frac{1}{2\zeta} \quad (3.19)$$

The viscous damping coefficient characterizes the resistance force (viscous damping force), in special vibration damping devices (shock absorbers, dampers, compensators, and the like). The viscous damping force is proportional to velocity. It is determined by the expression:

$$F_v = b \cdot \dot{u} \quad (3.20)$$

where b is the viscous damping coefficient.

\dot{u} is the velocity.

The various vibration damping devices are modeled in NX Nastran using *CVISC*, *CDAMPi*, *CBUSH*, and *CELASI* finite elements. Depending on the selected element type, the viscous damping coefficient value is specified in *PVISC*, *PDAMP*, *PBUSH*, *PBUSHT*, *PELAS* dialog boxes of the finite element physical property tables (Figure 3.4), or in relevant cards [7, 8]. The *CBUSH* element type is special because both viscous and structural damping coefficients can be specified.

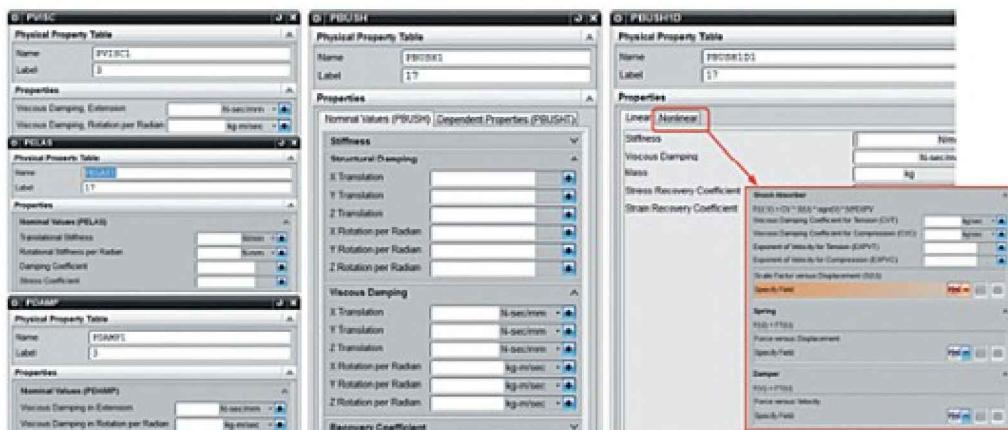


Figure 3.4. Specifying damping in *CVISC*, *CELASI*, *CDAMPi*, *CBUSH* elements

The structural damping coefficient is a dimensionless value that characterizes the force of structural (hysteretic) damping. The structural damping force is proportional to displacement and has a complex form:

$$F_s = i \cdot G \cdot k \cdot u \quad (3.21)$$

where G is the structural damping coefficient.

k is the stiffness.

u is the displacement.

$i = \sqrt{-1}$ is the imaginary unit, characterizing the phase shift.

There are two ways to specify the value of the structural damping coefficient:

- Specifying the corresponding value in the *Structural damping coefficient – GE* field in material properties (Figure 3.5) or in *MAT1* card in the *GE* box [7, 8].
- Assigning a structural damping coefficient to the *G* parameter in the *Solution Parameters* dialog box. You can also use the *PARAM,G,r* record in a NX Nastran file where r is the structural damping coefficient [7, 8]. Transforming structural damping into viscous damping to analyze unsteady-state oscillations also requires specifying reference frequencies using *PARAM,W3,r* or *PARAM,W4,r* parameters.

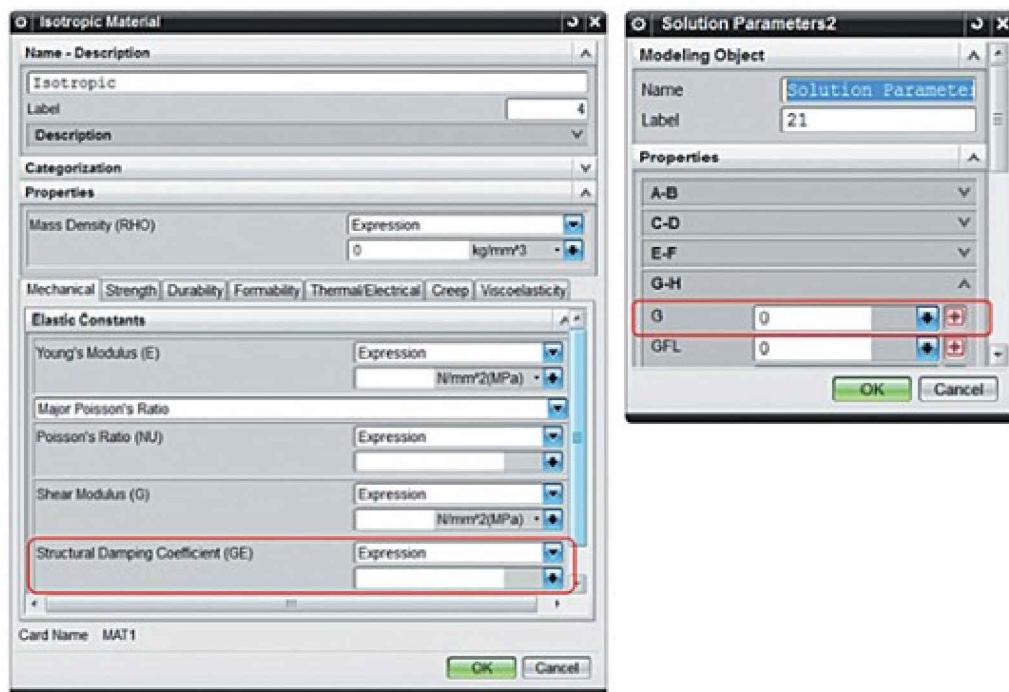


Figure 3.5. Specifying structural damping

Note that in a system with steady-state harmonic vibration, all parameters describing the damping of the system at a particular excitation frequency are related. At this frequency, structural and viscous damping forces are equal.

If this frequency is the natural angular frequency ω_n , then the following relations exist:

$$b = \frac{Gk}{\omega_n} = G\omega_n m \quad (3.22)$$

Using equations (3.18) and (3.5) for b_{cr} , the damping coefficient can be derived from structural damping:

$$\zeta = \frac{G}{2} \quad (3.23)$$

Then substituting (3.23) in (3.19) allows you to find the dynamic amplification factor in terms of structural damping using the following expression:

$$Q = \frac{1}{G} \quad (3.24)$$

These relations are useful for transforming one type of damping into another (viscous or structural) to specify dissipative properties of the system, because some NX Nastran solution types only allow the use of one specific damping type.

3.3. Determining natural frequencies and free vibration modes of structures

For most dynamics problems, determining natural frequencies and modes of the structure is recommended as an initial step. In terms of NX Nastran, this simulation type is known as the real eigenvalue analysis or normal modes analysis. *SEDCEIG 107 – Direct Complex Eigenvalues* and *SEMCEIG 110 – Modal Complex Eigenvalues* solutions are used for complex eigenvalue analysis.

It is known that the damping coefficient is small. For most structures it is 0 to 0.1 of the critical damping coefficient value. The ratio of the structure's natural frequency with damping to its frequency without damping is determined from expression (3.8), which shows that, for example, with damping coefficient of 0.1, the values of natural frequencies are virtually the same [7]. This explains the broad application of finding real frequencies of the structure in solving most dynamic problems, since accounting for

damping is difficult and computationally intensive. You can find real natural frequencies using *SEMODES 103*, *SEMODES 103 – Flexible Body*, *SEMODES 103 – Superelement* and *SEMODES 103 – Response Simulation* solutions.

Some of the reasons why finding natural frequencies and modes of a structure can be important are:

- Analyzing the interaction of a unit whose operation involves dynamic influence (oscillatory motion) and a system that acts as a support. In turn, it allows you to design the system so that the operating frequency of the device does not fall in the range near natural frequencies of the supporting structure. For example, in rotor-type machinery, this range is 0.7 through 1.3 of the natural frequency of the support [9]. The phenomenon of frequency coincidence is called resonance. It is characterized by a sharp increase of vibration amplitude, which causes the device to fail or damages the supporting structure.
- The obtained frequency values can inform design changes or evaluation of their effectiveness. For example, you can analyse the strain energy of finite elements of the simulation models at a particular mode to determine the regions of the structure where a change of stiffness can drive down the amplitudes of vibration or shift natural frequencies to prevent resonance.
- Analysis of natural frequencies is often used as a preliminary stage before various dynamic analyses. Results of this type of analysis can be used in subsequent modal decomposition dynamic analysis: modal transient analysis (*SEMTTRAN 112*), and modal frequency analysis (*SEMFREQ 111*). You can also use the eigenvalues to evaluate the appropriate time integration step for the equation of motion.
- Determination of optimal locations of vibration detectors (accelerometers) and vibration excitors when planning field experiments that involve gathering data or exciting modes in the frequency range of interest.

Real eigenvalues are determined by solving the equation of motion for the system with a finite number of degrees of freedom (3.17), neglecting

damping and external influence:

$$[M]\{\ddot{u}\} + [K]\{u\} = 0 \quad (3.25)$$

The solution of this equation that corresponds to harmonic oscillations at frequency ω , is of the following form:

$$\{u\} = \{\phi\} \sin \omega t \quad (3.26)$$

where $\{\phi\}$ is the eigenvector or mode.

Frequency ω and vector $\{\phi\}$ satisfy equation (3.27), which is equivalent to a system N of linear homogeneous equations with regard to components of eigenvector $\{\phi\}$:

$$([K] - \omega^2[M]) = 0 \quad (3.27)$$

or

$$\det([K] - \omega^2[M]) = 0 \quad (3.28)$$

Characteristic equation (3.28) is called the natural frequency equation. The number of positive roots of this equation is equal to the number of degrees of freedom of system N , which are called natural frequencies of that system. Each natural frequency ω_n has an eigenvector $\{\phi_n\}$. The ordered set of natural frequencies is the spectrum of natural frequencies of the system. Thus it can be concluded that for linear systems, dynamic behaviour of a linear elastic structure (deformation mode of linear elastic structure) can be described by a superposition of normal modes:

$$\{u\} = \sum \{\phi_n\} q_n = [\phi](q) \quad (3.29)$$

where q_n is the n th modal (generalized) coordinate.

Another notable property is the orthogonality of normal modes corresponding to different natural frequencies.

$$\{\phi_i\}^T [M] \{\phi_j\} = 0 \text{ при } i \neq j$$

$$\{\phi_i\}^T [M] \{\phi_j\} = m_j - j\text{-th modal (generalized) mass} \quad (3.30).$$

$$\{\phi_i\}^T [K] \{\phi_j\} = 0 \text{ npk } i \neq j$$

$$\{\phi_i\}^T [K] \{\phi_j\} = k_j - j\text{-th modal (generalized) stiffness (3.31).}$$

All modes are linearly independent, therefore, any system with a set of degrees of freedom can be represented as a set of unconnected systems with one degree of freedom (Figure 3.6).

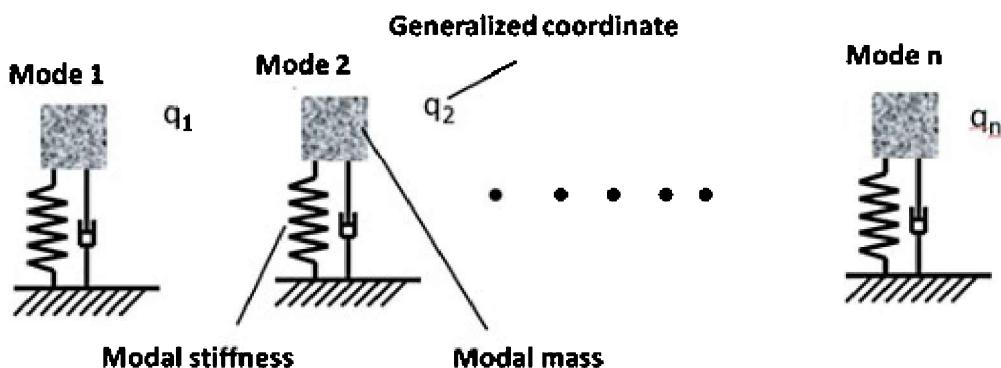


Figure 3.6. A set of systems with one degree of freedom

NX Nastran uses seven approximatation methods to calculate eigenvalues. The choice of a suitable, most efficient method depends on a number of considerations, for example, the size of the simulation model, the number of unknown eigenvalues, available RAM, and the degree of singularity of the mass matrix. There are two groups of algorithms that can be used to find eigenvalues: iterative methods, and similarity transformation methods also known as reduction methods.

The first group includes the following methods:

- Inverse power method
- Modified inverse power method

The second group includes the following methods:

- Givens method
- Modified Givens method
- Householder method
- Modified Householder method

The Lanczos method stands apart because it combines all advantages of

iterative methods and reduction methods.

Iterative methods are useful and efficient for determining the smallest and largest eigenvalues, however, some modes can be lost.

Reduction modes are more complicated and computation-intensive, but they allow the determination of natural frequencies and modes without losing any.

Nevertheless, the Lanczos method is recommended for most problems. In **NX Advanced Simulation**, the Lanczos method is used by default. The most important prerequisites for this method are: non-negative mass matrix and symmetric stiffness matrix. The Lanczos method does not miss eigenvalues and has all the advantages of iterative methods. Note that in contrast to other methods, the Lanczos method undergoes constant development to improve its performance, so it is recommended for modal analysis of medium and large simulation models. For more details on methods available in NX Nastran, refer to [7].

It is notable that you can find eigenvalues and free vibration modes with the pre-stressed state of the structure taken into account. This can cause the stiffness matrix to change. In this case the solution involves two steps. The first step is linear static analysis of the structure. This is achieved using *Step – Subcase – Static*. The second step is finding natural frequencies and modes of the pre-stressed structure using *Step – Subcase – Eigenvalue Method*.

You can also find natural frequencies and free vibration modes while accounting for cyclical symmetry. In this way, you can significantly reduce the dimensionality of a cyclically symmetric model.

Example 1. Determining natural frequencies and free vibration modes

The following is a calculation of natural frequencies and free vibration modes using an FE model of a tower. All preliminary stages, including preparation and idealization of the CAD model, FE mesh creation, specifying material properties, and physical properties of finite elements are already completed. This example demonstrates how to create a solution for finding eigenvalues, specify the required solution parameters, set up output options, and view

results.

1. Opening a simulation model

Launch NX and open the *Tower_sim1.sim* file. You automatically switch to **NX Advanced Simulation**. The FE model of the tower consists of CQUAD4 2D elements and stiff *RBE2* elements used to link nodes of the base of the tower. Note that the model tree in the Simulation Navigator tab of the Resource Bar contains only 1D and 2D collectors of the FE mesh and does not contain solutions or containers of boundary constraints and simulation objects (Figure 3.7).

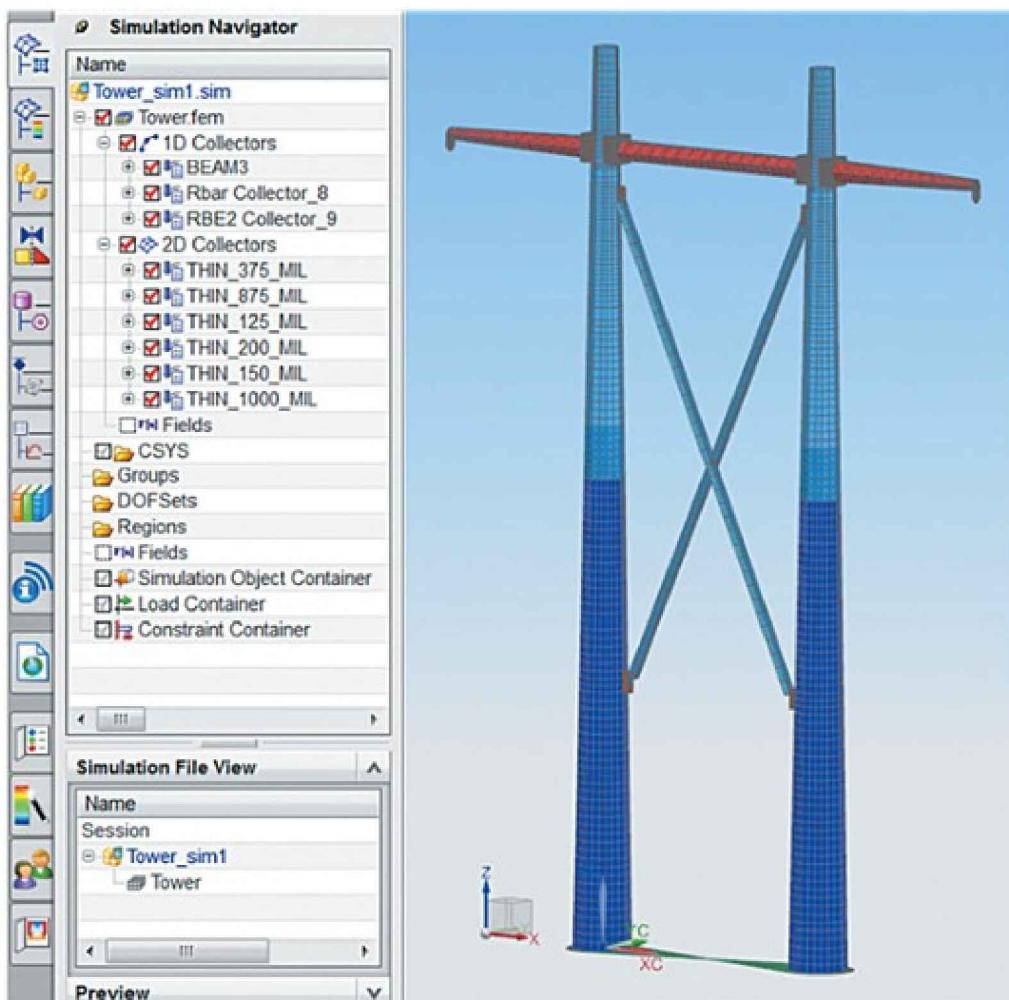


Figure 3.7. FE model of a tower

Reset all dialog box settings to default by choosing the following main menu command: **Preferences**→**User Interface**→**General**→**Reset Dialog Box**

Settings, and clicking the **OK** button.

2. Creating a new solution for finding natural frequencies and normal modes

In the model tree of the Simulation Navigator tab, right-click *Tower_sim1.sim* (the first item in the model tree) and choose *New Solution*. In the *Solution* dialog box (Figure 3.8), in the *Solution Type* list, select *SEMODES 103*. Change the solution name to *Modal_Analysis*, click **OK**.

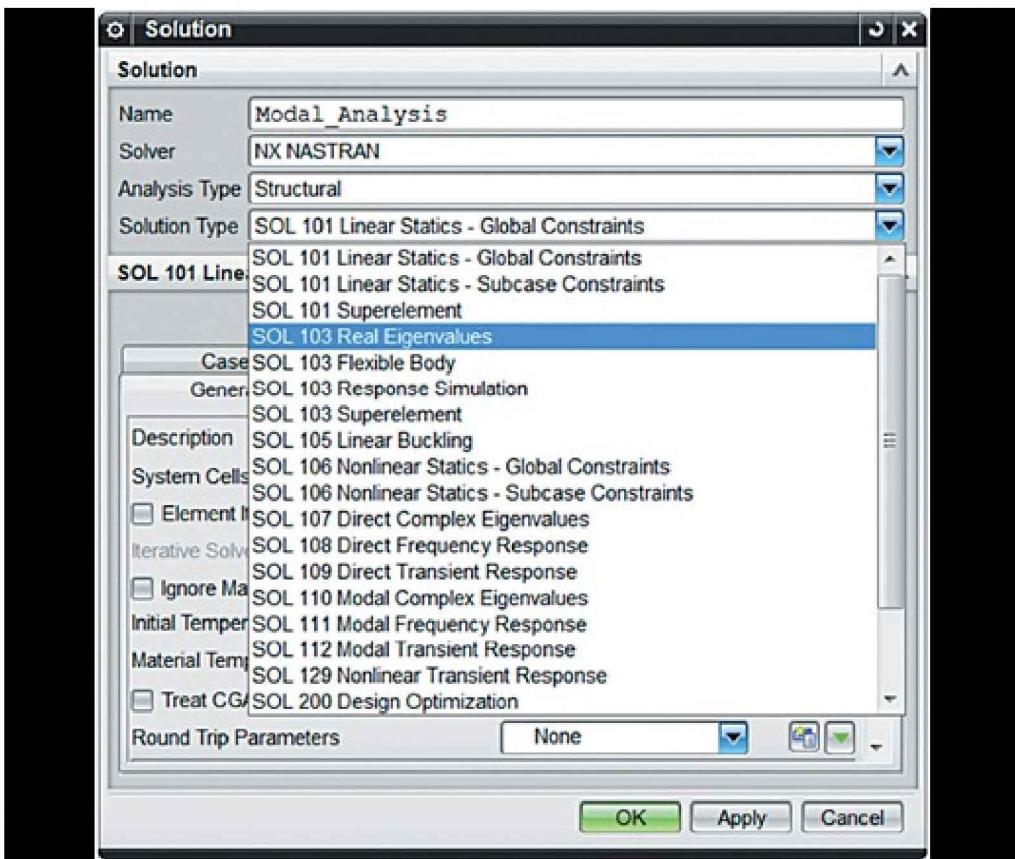


Figure 3.8. Creating a new solution

3. Specifying solution parameters

In the **Solution Step** (Figure 3.9) dialog box, make sure you select *Step – Subcase – Eigenvalue Method*.

Select the Lanczos method for finding eigenvalues. The interface of NX Advanced Simulation provides two eigenvalue calculation methods, Lanczos and Householder, however, the first of the two methods is recommended. You can specify a different method by editing the *EIGR* card

of NX Nastran input file [8].

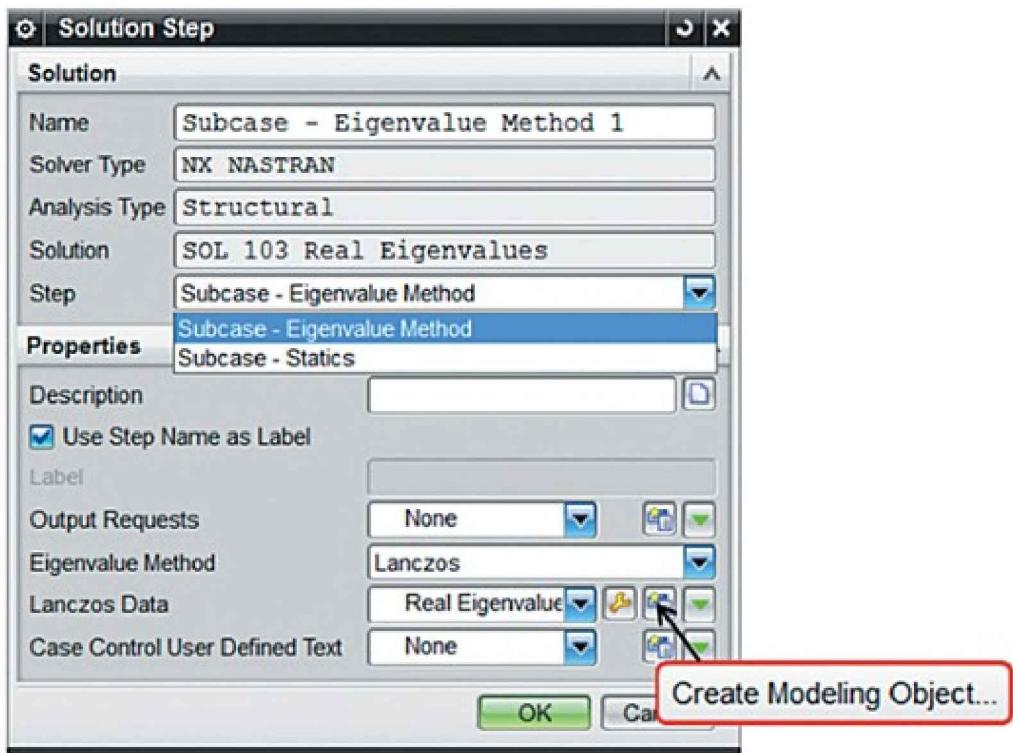


Figure 3.9. Specifying the eigenvalue finding method

In the **Solution Step** dialog box, next to the *Lanczos Data* option, click **Create Modeling Object**. In the **Real Eigenvalue – Lanczos1** dialog box, you can specify the following parameters (Figure 3.10):

- In the *Frequency Range – Lower Limit/Upper Limit* boxes, you can enter the required range for finding natural frequencies. Enter an upper limit of 100 Hz.
- In the *Number of desired Modes* box, you can type the number of natural frequencies and modes to find.
- In the *Number of Vectors* box, you can type the number of vectors in the Lanczos method block. A value of 7 is suitable for most models. Using a larger number can speed up computation for large problems.
- In the *Estimate of the First Natural Frequency* box, you can enter a known value of the first natural frequency to speed up computation for Lanczos method.

- In the *Method for Normalizing Eigenvectors* list, you can select:
 - MASS* to use the default normalization method for normal mode analysis. It reduces the generalized mass value to unity.
 - MAX* to use the normalization method that reduces to unity the largest displacement value for degrees of freedom [7, 8]. All eigenvectors are normalized with respect to this largest value, which is useful for determining the relative contribution of each vibration mode.

In this example, keep default values for all parameters except the upper limit of the search range (Figure 3.10). Click **OK**. The **Solution Step** dialog box stays open.

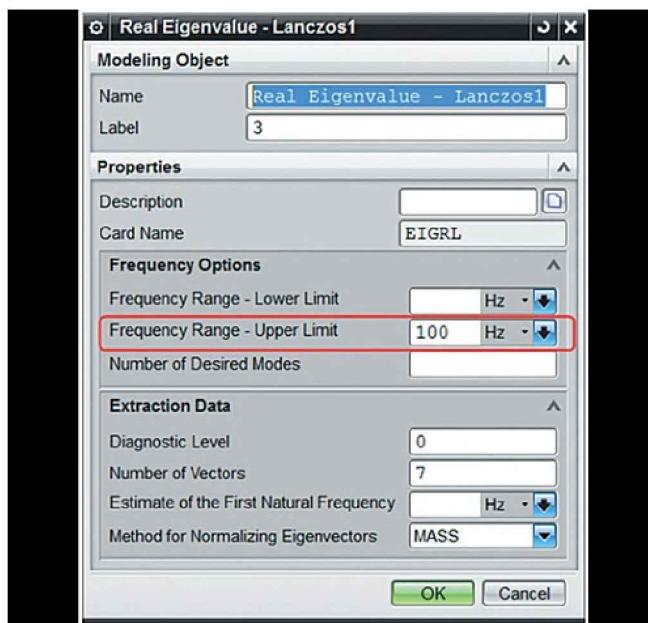


Figure 3.10. Specifying parameters for finding eigenvalues

4. Specifying result output options

In the **Solution Step** dialog box, next to the *Output Requests* option, click **Create Modeling Object**. In the **Structural Output Requests1**, specify the following parameters (Figure 3.11):

- In the *Displacement* tab, make sure the *Enable DISPLACEMENT Request* check box is selected.
- In the *Modal effective mass* tab, specify the following:
 - Select the *Enable MEFFMASS request* check box.

- b. In the *Modal participation factor* list, select the *PARTFAC* option for output into the results file.
- c. In the *Modal effective mass* list, select the *MEFFM* option for output into the results file in units of mass (alternatively, select *MEFFW* for output in units of weight).

In the **Structural Output Requests1** dialog box, click **OK**.

In the **Solution Step** dialog box, click **OK**.

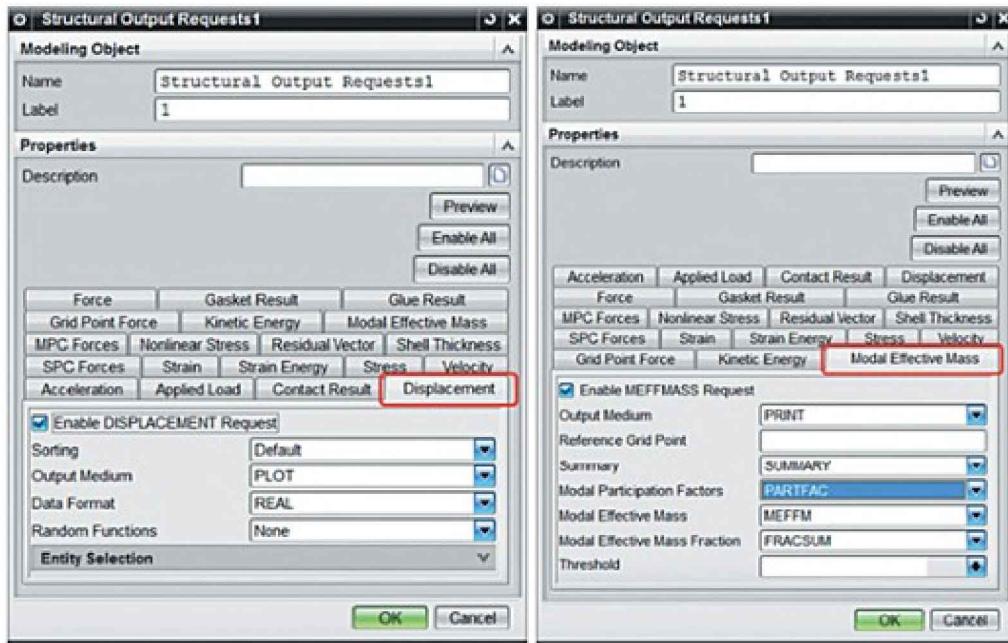


Figure 3.11. Specifying result output options

Note that in the model tree of the Simulation Navigator tab, there is a new *Modal_Analysis* node corresponding to the created solution.

5. Specifying constraints

To specify degree of freedom constraint conditions for the structure, in the model tree of the Simulation Navigator tab, right-click Constraint container to display the shortcut menu. In the drop-down list, choose *Fixed Constraint* (constrain all degrees of freedom). In the *Fixed Constraint* dialog box (Figure 3.12), select node 20000 (independent node of *RBE2* elements), and click **OK**.

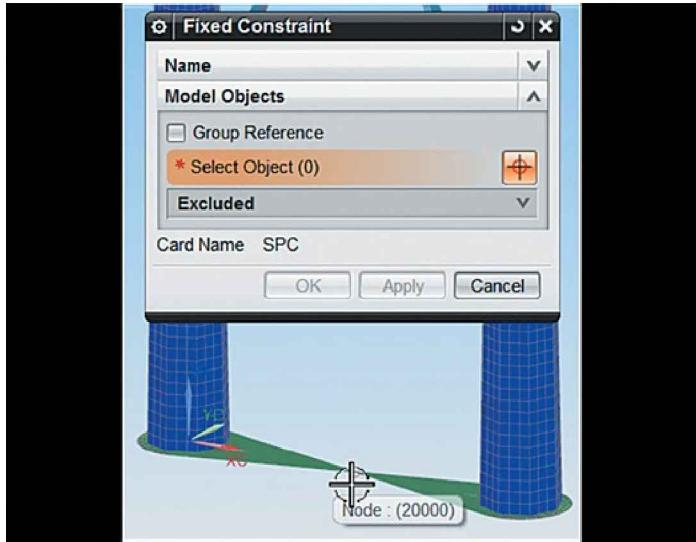


Figure 3.12. Specifying constraints

The new *Fixed(1)* constraint is saved in the *Constraints* node of the active *Modal_Analysis* solution.

The simulation model is ready for calculation. To save the model, choose **File→Save** in the main menu.

6. Running the solution

To run the solution, right-click *Modal_Analysis* and choose **Solve...** in the shortcut menu. In the **Solution** dialog box, click **OK**. The computation can take several minutes.

The **Solution Monitor – tower_sim1-modal_analysis** dialog box automatically opens (Figure 3.13). It contains three tabs:

- *Solution Information*.
- *Sparse Matrix Solver*, which shows the number of computed equations.
- *Eigenvalues Extraction*, which shows the number of computed eigenvalues.

The computation is complete when you see the *Job Finished* message.

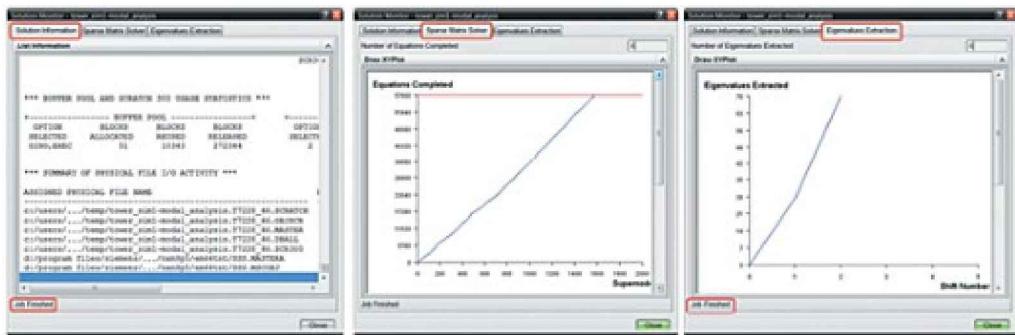


Figure 3.13. Solution Monitor

7. Viewing simulation results

7.1. In the **Simulation Navigator** model tree, double-click *Results* and select the **Post Processing Navigator** tab, which contains the loaded results. To view the results of modal analysis, expand the *Modal_Analysis* node, select the frequency of interest, for example, the third (*Mode 3*), expand its node and double-click *Displacement–Nodal* to display the deformed state corresponding to the third free vibration mode in the graphics area (Figure 3.14).

7.2. Normal vibration modes at particular frequencies are typically displayed in animated form. To display animation, on the **Post Processing** toolbar, click the **Animation command**. Set the following options in the dialog box (Figure 3.14):

- In the *Animate* list, select *Result*.
- In the *Style* list, select *Modal*.
- In the *Number of Frames* box, type 10.
- Select the *Full-cycle check box*.
- Click **OK**.

The result is an animated representation of the structure vibrating at the third natural frequency.

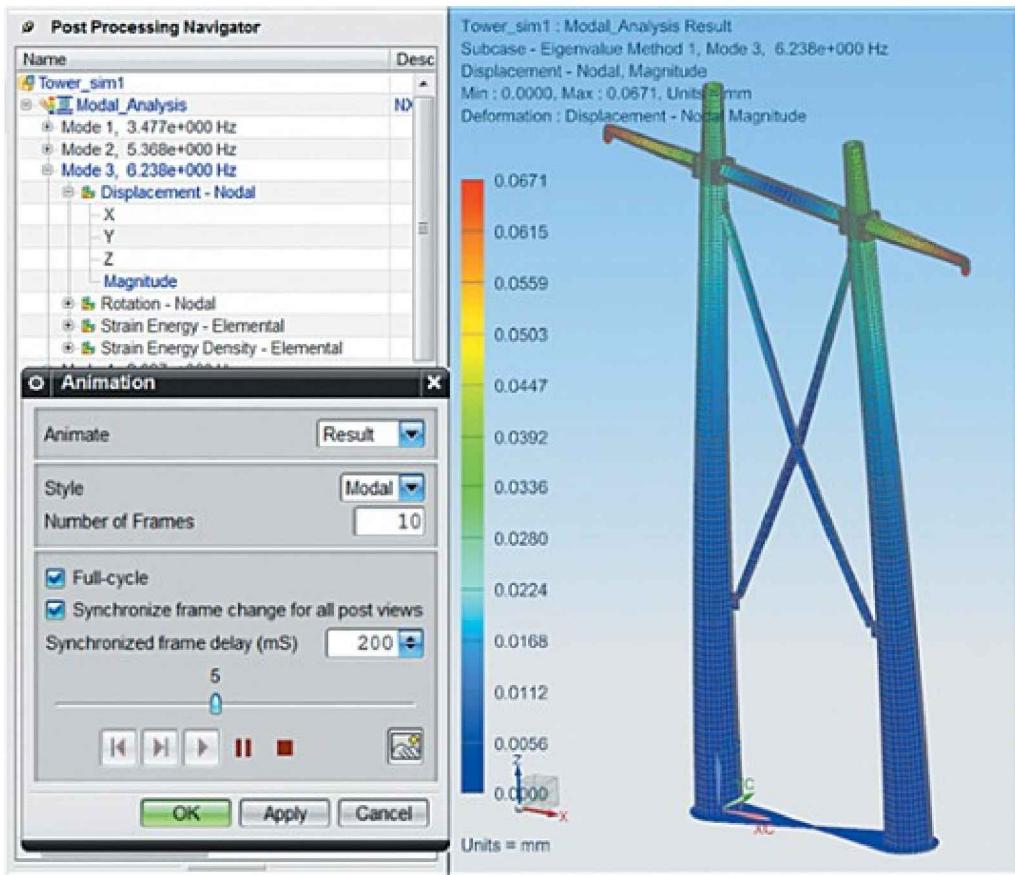


Figure 3.14. Viewing simulation results

In the working folder, open the *tower_sim1-modal_analysis.f06* file in a text editor. Find the tables titled *Modal effective mass* and *Modal participation factor*. Each of these tables has 70x6 cells, where 70 is the number of eigenvalues found, and 6 is the number of degrees of freedom.

Each row of the *Modal effective mass* (Figure 3.15) table characterizes the share of effective modal mass for each vibration mode, while each cell of the row characterizes the share of effective modal mass for a degree of freedom. For example, for the third mode shown in Figure 3.14, main displacements correspond to rotation around the OZ axis, therefore the sixth component of the third row has the largest value.

MODAL EFFECTIVE MASS							
MODE NO.	FREQUENCY	T1	T2	T3	R1	R2	R3
1	3.477512E+00	2.729001E-03	7.038203E+00	1.362996E-07	8.648078E+05	3.356327E+02	3.057186E+04
2	5.368261E+00	8.135398E-01	6.889125E-04	2.074111E+03	9.869156E+04	2.110691E+00	
3	6.339010E+00	9.645553E-07	8.6862498E-12	2.201531E-06	3.210998E-06	7.672807E-02	4.031133E+03
4	7.852010E+00	7.266894E+00	5.366834E-03	2.744314E-08	1.765982E+01	8.184821E+05	2.718988E+01
5	1.275313E+01	1.419659E+05	4.462110E-07	7.794838E-03	3.628582E-03	1.546217E+01	1.725593E+02
6	1.347751E+01	1.347751E+05	5.314144E-03	2.744314E-08	1.765982E+01	8.184821E+05	2.718988E+01
7	1.413334E+01	1.619313E+05	7.407102E-08	5.083724E-03	1.925038E-03	3.074868E+01	2.455691E+00
8	1.451038E+01	5.951105E-03	1.015854E-02	2.710988E-06	1.180491E-02	1.088160E+03	4.524450E+01
9	1.486211E+01	5.508640E-06	1.583794E-05	2.010193E-02	1.034352E+01	5.517000E+01	
10	1.657917E+01	4.607231E-01	1.009909E-03	1.767280E-08	1.191179E+01	3.603122E+02	4.613762E+00
11	1.907206E+01	5.216335E-06	2.414643E-07	2.1955569E-03	1.444733E+01	8.817817E+01	
12	1.969161E+01	2.624341E-06	2.675178E-06	5.192438E-01	9.451711E-03	2.275028E+03	3.294498E+00
13	2.094176E+01	4.002534E-01	1.719904E-04	7.919889E-07	1.711480E+00	1.340042E+00	4.678184E-01

Figure 3.15. A segment of the Modal effective mass table

The *Modal participation factor* table (Figure 3.16) shows the modal participation coefficient of each degree of freedom for a particular vibration mode. For the third vibration mode (the third row of the table), the sixth component is the most important.

MODAL PARTICIPATION FACTORS							
MODE NO.	FREQUENCY	T1	T2	T3	R1	R2	R3
1	3.477512E+00	5.223984E-02	-2.652961E+00	-3.691877E-01	9.299504E+02	1.832028E+01	-1.748481E+02
2	5.368261E+00	-9.019589E-01	2.624714E-02	-6.546253E-05	4.554243E+01	-3.141553E+02	1.452822E+00
3	6.339010E+00	-9.821177E-01	-2.947244E-06	-1.4887616E-03	1.791926E-03	-2.769947E-01	-2.007688E+00
4	9.987703E+00	2.695736E+00	7.2604185E-02	1.656597E-04	-4.202358E+00	9.047000E+02	5.2124391E+00
5	1.275211E+01	5.847786E-03	-6.688738E-04	8.828838E-02	6.022715E-02	-3.932323E+00	1.313717E-01
6	1.347751E+01	-1.321010E+00	8.610102E-02	1.723559E-03	-8.640400E+00	-2.848510E+01	5.555556E+00
7	1.413334E+01	-8.462324E-03	7.723599E-04	7.130024E-02	-4.387525E-02	-5.544494E+00	-1.567064E+00
8	1.451038E+01	7.714339E-02	-1.017770E-01	-1.100449E-03	-1.086505E+01	3.298728E+01	6.726403E+00
9	1.486211E+01	2.347049E-03	-1.258489E-03	-3.884072E-02	1.417813E-01	3.2161344E+00	-7.427651E-01
10	1.657917E+01	6.787658E-02	3.176625E-02	-1.129191E-04	-3.451346E+00	-1.898189E+01	2.147967E+00
11	1.907206E+01	2.283930E-03	-4.913902E-04	6.999504E-02	4.685690E-02	-3.800967E+00	9.390324E+00
12	1.969161E+01	1.619982E-03	1.6305597E-03	-7.205857E-01	-9.721991E-02	4.769725E+01	1.815075E+00
13	2.094176E+01	-6.326558E-01	1.311458E-02	-8.899376E-04	-1.308235E+00	3.660659E+01	6.839725E-01

Figure 3.16. Segment of the Modal participation factor table

After analysing the obtained results, save the model. To do so, choose **File→Save all** in the main menu. Close the simulation model: **File→Close→All Parts**.

3.4. Frequency response analysis

Frequency response analysis involves finding the steady-state reaction of the structure to harmonic excitation. In a harmonic excitation scenario, all external forces must be explicitly defined for all excitation frequencies. In a steady-state vibration process, all degrees of freedom typically experience motion with the same frequency. As previously noted, the system's response may be out of phase with the external influence. The phase shift is due to damping. Results of frequency response analysis are complex numbers. You can output either amplitude and phase, or real and imaginary components. Complex displacements are of the following form:

$$u(\omega) = u_0 (\cos\varphi + i\sin\varphi) \quad (3.32)$$

where u_0 is the amplitude of displacement.

$u_0 \cos\varphi$ is the real component of the displacement vector.

$u_0 \sin\varphi$ is the imaginary component of the displacement vector.

You can solve frequency response analysis in NX Nastran using one of the following two methods: the direct method, using the *SEDFREQ 108 (Direct Frequency Response)* solution, or the modal method, using the *SEMFREQ 111 (Modal Frequency Response)* solution. The choice of the method depends on the size of the system being studied, the required precision of results, the number of excitation frequencies, the need to analyse the high-frequency region, and other considerations.

3.4.1. The direct integration method

The direct integration method determines the system's response at each particular excitation frequency by solving equations of motion using complex algebra. The equation of motion for steady-state forced vibrations of the system is of the following form, considering the system's dissipative properties:

$$[M]\{\ddot{x}(t)\} + [B]\{\dot{x}(t)\} + [K]\{x(t)\} = \{P(\omega)\}e^{i\omega t} \quad (3.33)$$

The external load vector in the right-hand part of the equation is complex, just like the system's reaction vector. Considering (3.32), the solution of equation (3.33) is:

$$\{x\} = \{u(\omega)\}e^{i\omega t} \quad (3.34)$$

By substituting the first-order and second-order time derivatives of displacement into (3.33) and cancelling $e^{i\omega t}$, the following expression is obtained:

$$[-\omega^2 M + i\omega B + K]\{u(\omega)\} = \{P(\omega)\} \quad (3.35)$$

To find the response of the structure, you need to solve the equation of motion for each particular excitation frequency ω (that is, a set of equations with different excitation frequencies) in a way similar to solving equations of static equilibrium using complex algebra.

NX Nastran provides six parameters for choosing frequencies for searching a solution [7, 8]:

FREQ specifies discrete excitation frequencies.

FREQ1 specifies a set of excitation frequencies by specifying the initial frequency, the frequency increment and the number of frequency increments.

FREQ2 specifies a set of excitation frequencies by specifying the initial frequency, the final frequency, and the number of logarithmic increments.

FREQ3 specifies the number of excitation frequencies between two natural frequencies in the range between the initial and the final frequencies.

FREQ4 specifies a set of excitation frequencies by setting the initial and the final frequencies, a range in the vicinity of each natural frequency, and the number of uniformly distributed frequencies in that range.

FREQ5 specifies a set of excitation frequencies by setting the initial and the final frequency, and a set of fractions of the system's natural frequencies.

In the first method of frequency response analysis, the damping of the system is represented by matrix $[B]$, which characterizes viscous damping of some types of finite elements *CVISC*, *CDAMPi*, *CBUSH*, *CELASI*, and complex stiffness matrix $[K]$, which includes both the stiffness of the system and the structural damping. For the purposes of frequency response analysis, structural damping as specified by the **PARAM,G** parameter and the structural damping coefficient of the material **GE**, does not directly define the damping matrix, instead forming the global stiffness matrix:

$$[K] = (1 + iG)[K] + i \sum G_E [K_E] \quad (3.36)$$

where $[K]$ is the global stiffness matrix.

G is the total structural damping coefficient (**PARAM,G**).

$[K_E]$ is the elemental stiffness matrix.

G_E is the elemental structural damping coefficient (**GE**).

Structural damping coefficients are automatically added to the stiffness matrix and accounted for in the equations of motion.

Note that a solution can use all damping types. In contrast to transient analysis, you can use a complex stiffness matrix to make the transformation of structural damping to equivalent viscous damping unnecessary.

Example 2. Direct integration frequency response analysis

Direct-integration frequency response analysis can be demonstrated using

the FE model of the tower from Example 1.

Launch NX and open the *Tower_sim1.sim* file. You automatically switch to NX Advanced Simulation.

Reset all dialog box settings to default by choosing the following main menu command: **Preferences**→**User Interface**→**General**→**Reset Dialog Box Settings**, and clicking the **OK** button.

Note that the model tree of the Simulation Navigator already has the *Modal_Analysis* tab corresponding to the solution carried out in Example 1.

1. Creating a group of elements and nodes

Create a group of elements and nodes, and set up results output into an *.op2* file. In the model tree of the Simulation Navigator tab, right-click the *Groups* node and select *New group*. In the dialog box (Figure 3.17), specify the following:

- In the *Element Labels* box, type element numbers 3539 and 8232.
- In the *Node Labels* box, type node numbers 4076, 4370, 8471, 8885.
- In the *Name* box, keep the default name *Group(1)*.

Click **OK**.



Figure 3.17. Creating a group of elements and nodes

2. Creating a new solution for direct frequency response analysis

In the model tree of the Simulation Navigator tab, right-click *Tower_sim1.sim* (the first item in the model tree), and choose *New Solution*. In the *Solution* dialog box (Figure 3.18), in the *Solution Type* list, select *SEDFREQ 108*. Change the name of the solution to *Direct_frequency_response*. The *Solution* dialog box stays open.

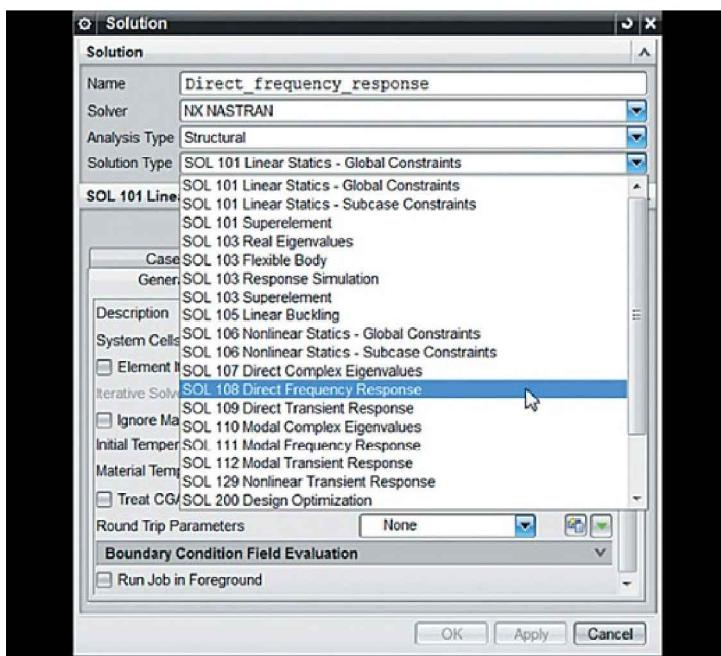


Figure 3.18. Creating a new solution

3. Specifying result output options

In the *Solution* dialog box, select the *Case Control* tab and next to the *Output Requests* option, click **Create Modeling Object**. In the *Structural Output Requests2* dialog box (Figure 3.19), click *Disable All* and set the following options:

- In the *Displacement* tab:
 - a. Select the *Enable DISPLACEMENT Request* check box.
 - b. In the *Sorting* list, select *SORT2* to enable working with the *.op2* results file in the **XY Function Navigator** tab.
 - c. In the *Object* list, select *Group*.
 - d. In the *Group* list, select the previously created group *Group(1)*.
- In the *Stress* tab:
 - a. Select the *Enable STRESS Request* check box.

- In the *Sorting* list, select **SORT2**.
- In the *Object* list, select *Group*.
- In the *Group* list, select the previously created group *Group(1)*.

In the **Structural Output Requests2** dialog box, click *Preview*. The **Information** dialog box appears with information on the data queried for writing into the results file.

In the **Structural Output Requests2** dialog box, click **OK**.

The **Solution** dialog box stays open.

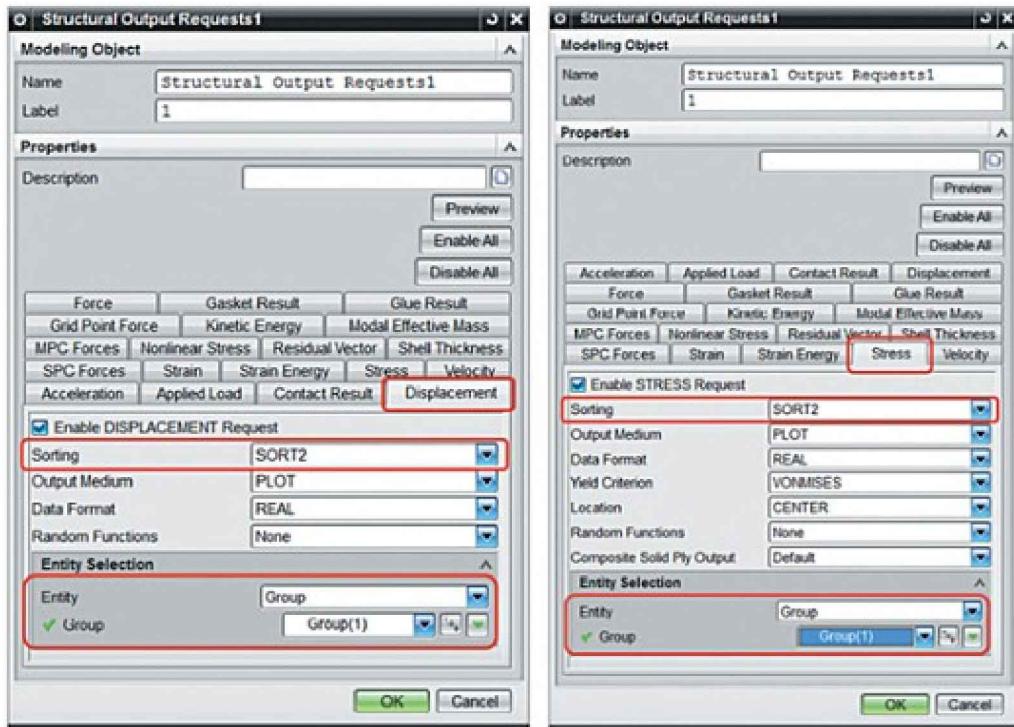


Figure 3.19. Specifying results output options

4. Specifying the structural damping coefficient G

In the **Solution** dialog box, in the **Parameters** tab, click **Create Modeling Object**. In the **Solution Parameters1** (Figure 3.20), select the **G-H** tab, in the **G** box, type the value of the structural damping coefficient (**0.08**), then click **Add**. Click **OK** in all dialog boxes.

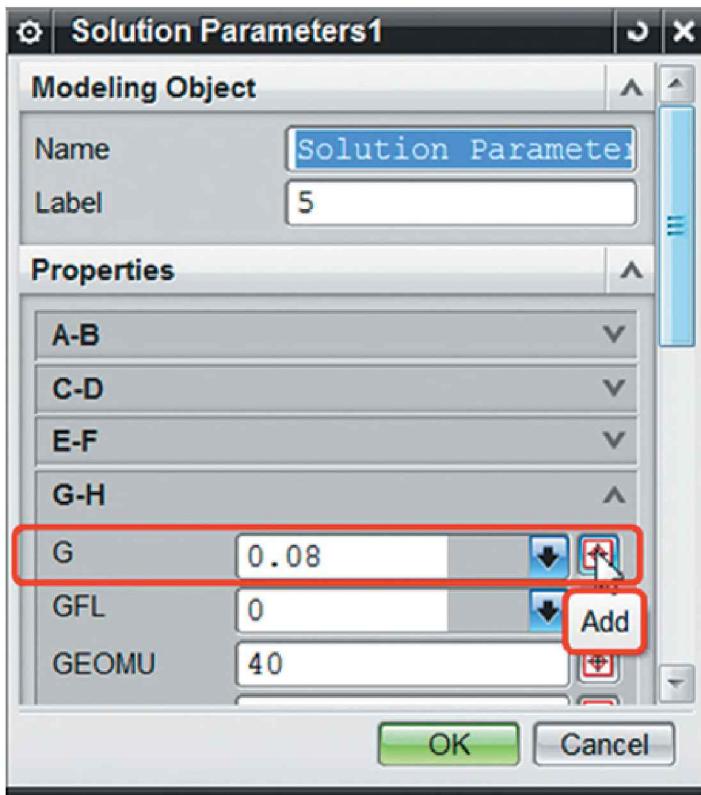


Figure 3.20. Specifying the structural damping coefficient G

Note that in the model tree of the Simulation Navigator tab, there is a new *Direct_frequency_response* node corresponding to the solution you just created.

5. Specifying loads and boundary constraints

5.1. To specify degree of freedom constraints, in the Simulation Navigator tab, expand the *Constraint container* node of the tree view and drag the *Fixed(1)* constraint to the *Constraints* node of the *Direct_frequency_response* solution (Figure 3.21).

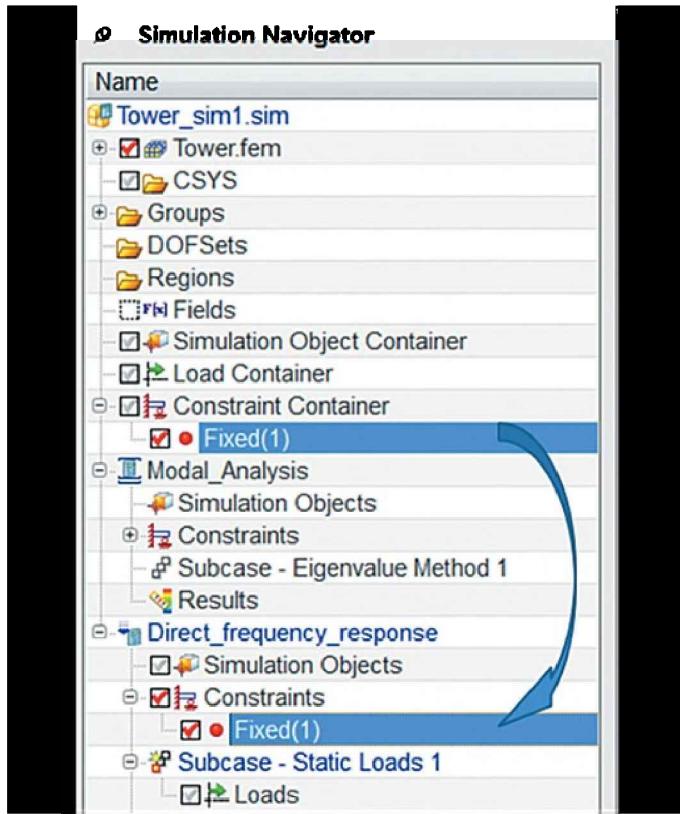


Figure 3.21. Specifying the *Fixed(1)* constraint

5.2. Specify wind load as the external load. The value calculated in accordance with the Russian building code (SNiP 2.01.07–85* “Loads and effects”) is 2.1 kPa . Assume that the wind direction coincides with the positive direction of the OY axis of the global coordinate system. The range of the impact is 1 through 10 Hz. The pressure distribution along the tower is non-uniform, linearly increasing from 0 through 1 with a slope of $9.1E-5$ and a scaling factor of 2100.

To create wind load pressure, in the Simulation Navigator model tree, right-click *Load container*, and in the **New Loads** submenu of the shortcut menu, choose *Pressure*. In the dialog box, specify the following (Figure 3.22):

- In the *Type* list, select *Components*.
- Under *Select Object*, select elements oriented along the OY axis with regard to the XOZ plane, as shown in Figure 3.22. For convenience, you can use *Selection Filter – Element Face*, *Selection Method – Feature Angle Element Face*.

- In the *Magnitude* list, select *Field*.
- Under *Define Field*, select *Table Constructor*. In the *Table Field* dialog box (Figure 3.23), specify the following:
 - a. In the *Domain* list of the *Independent* tab, select *Frequency*.
 - b. In the *Data Points* group, type “1,0,1,0” in the data entry box, then click *Accept edit*.
 - c. In the *Data Points* group, type “10,0,1,0” in the data entry box, then click *Accept edit*.
 - d. Click **OK**.
- Under *Scale factors*, specify $P_y = 2100$.
- In the *Distribution* group, select *Spatial* in the *Method* list.
- In the *Distribution field* list, select *Formula constructor*. In the **Formula Field** dialog box (Figure 3.23), specify the following:
 - a. In the *Domain* list of the *Independent* tab, select *Cartesian*.
 - b. In the *Expressions* group, type “9.1E-5” in the data entry box.
 - c. In the *Filters* box, double-click variable Z. Its internal NX designation *ug_var(“z”)* is inserted into the data entry box. The full expression is “9.1E-5**ug_var(“z”)*”. Click *Accept edit*.
 - d. In the **Formula Field** dialog box, click **OK**.
- In the *Pressure* dialog box, click **OK**.

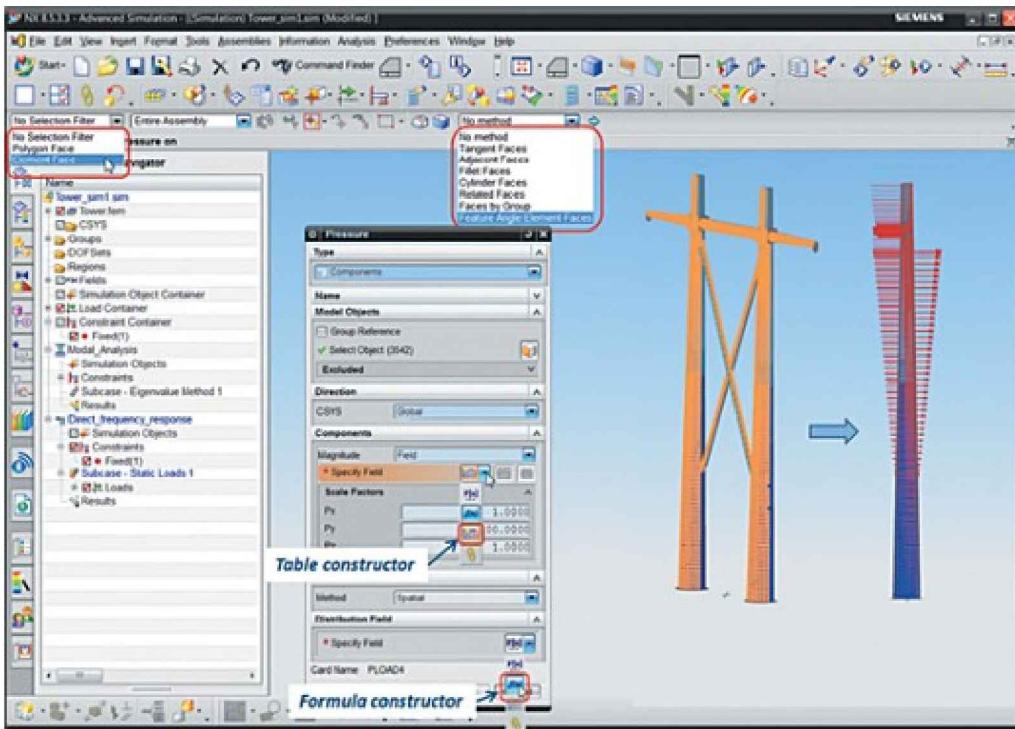


Figure 3.22. Specifying pressure

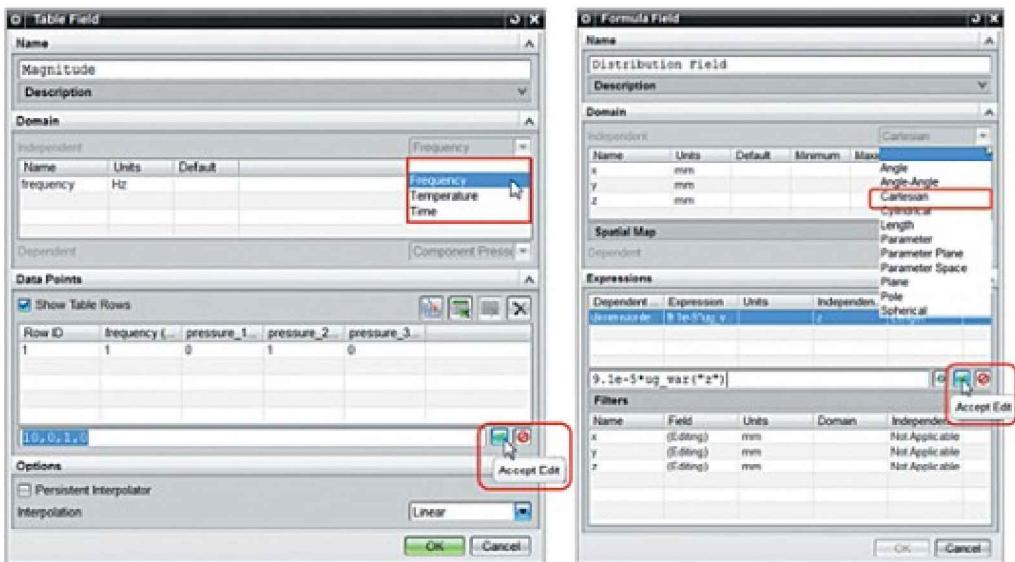


Figure 3.23. Specifying distribution using a formula

The *Pressure(1)* load you just created is added to the *Loads* node of the active solution (*Direct_frequency_response*).

6. Specifying excitation frequencies

In frequency response analysis, the solution is carried out for a particular excitation frequency. To specify excitation frequencies, select the Simulation Navigator tab and right-click the *Subcase – Direct Frequency 1* node that corresponds to the step of the frequency response analysis solution, then choose *Edit*. The *Solution Step* dialog box appears. Change the name to *Direct Frequency*. Under *Forcing Frequencies*, click **Create Forcing Frequencies**. In the **Modeling Object Manager** dialog box, click *Create*. In the **Forcing Frequencies – Direct1** dialog box, specify the following (Figure 3.24):

- In the *Frequency List Form* list, select *FREQ1*.
- In the *First Frequency* box, enter 1 Hz.
- In the *Frequency Increment* box, enter 0.2 Hz.
- In the *Number of Frequency Increments* box, type 45.

Click **OK**.

In the new **Modeling Object Manager** dialog box, do the following:

- Click *Add to list*.
- Click *Close*.

Click **OK** in all dialog boxes.

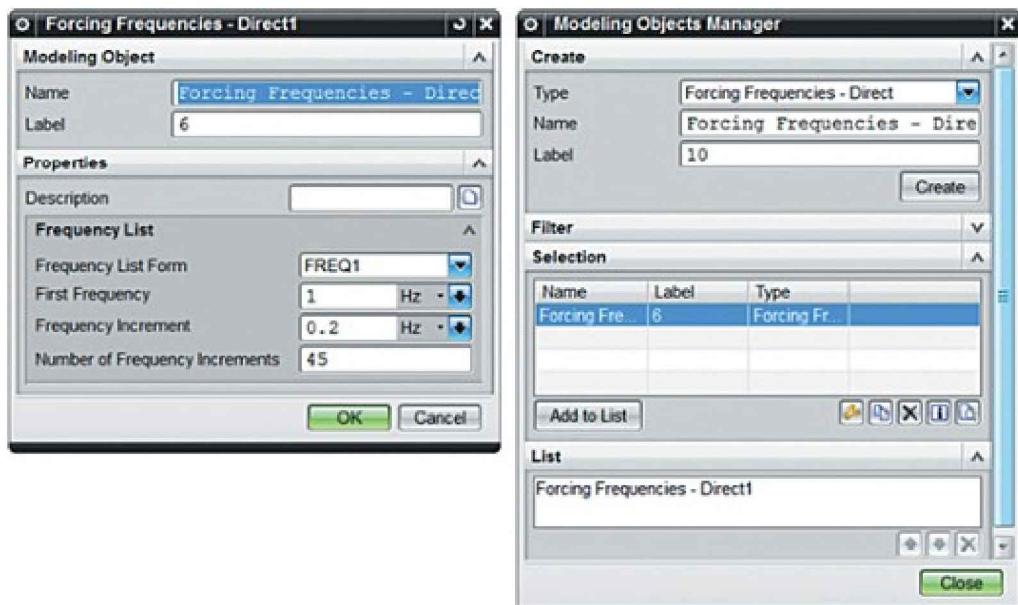


Figure 3.24. Specifying excitation frequencies

The simulation model is ready for calculation. To save the model, choose **File→Save** in the main menu.

7. Running the solution

To run the solution, right-click *Direct_frequency_response* and choose **Solve...** in the shortcut menu. In the *Solution* dialog box, click **OK**. The computation can take several minutes.

When the computation is complete, close all new windows.

8. Viewing simulation results

Go to the **XY Function Navigator** tab of the Resource Bar. Right-click the *Nastran OP2 Files-Sort2 Results* node and choose *Open* in the shortcut menu. Specify the *tower_sim1_direct_frequency_response.op2* results file. With the file opened in the **XY Function Navigator** tab, expand the *tower_sim1_direct_frequency_response* node, then expand the *Displacement (Subcase 1)* node. Hold *Ctrl* and click *Node 4370: Ty* and *Node 8885: Ty* to select them, then right-click and choose the *Plot XY* command. The graphics area shows the frequency response of the two nodes on the same graph (Figure 3.25). Use commands of the **XY Graph** toolbar to edit the graph as necessary (refer to Chapter 5, Part 1).

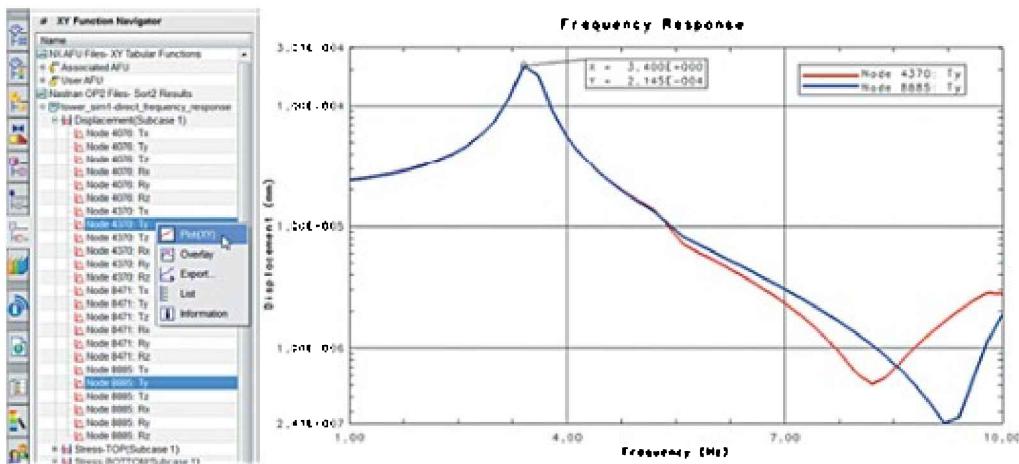


Figure 3.25. Viewing simulation results

Working independently, display results for other nodes.

3.4.2. The modal method

The modal method (mode superposition method) is an alternative method of

finding a structure's response to dynamic influence. This method uses pre-determined natural frequencies and free vibration modes. The method is based on the ability to represent a deformed state of a linear elastic system with a set of its normal vibration modes, that is, to go from physical coordinates $\{u(\omega)\}$ to modal (generalized) coordinates $\{\xi(\omega)\}$ [7].

$$\{x\} = [\phi]\{\xi(\omega)\}e^{i\omega t} \quad (3.37)$$

If damping is neglected, and the orthogonality property of normal vibration modes is respected, then in terms of generalized coordinates, the equation of motion is of the following form:

$$-\omega^2 m_i \xi_i(\omega) + k_i \xi_i(\omega) = p_i(\omega) \quad (3.38)$$

where in accordance with equations (3.30) and (3.31), m_i is the i -th modal (generalized) mass.

k_i is the i -th modal (generalized) stiffness.

p_i is the i -th modal (generalized) force.

Therefore, the equation of motion is replaced with a set of decoupled equations, each of which is an equation of motion for a system with one degree of freedom. By solving this system of equations, you can obtain a generalized reaction $\xi_i(\omega)$ for each of the modes. Correspondingly, after finding all generalized coordinates for all relevant modes, you can determine the physical displacement vector by summing up all modal response reactions of the system:

$$\{x\} = \sum \{x_i\} = \sum [\phi_i]\{\xi_i(\omega)\}e^{i\omega t} \quad (3.39)$$

With the modal method of frequency response analysis, both the damping matrix $[B]$ and the global complex stiffness matrix are not diagonal, therefore, the independence condition of modal equations of motion is not satisfied. In this case, the system of coupled equations is solved by numerical integration, in a way similar to the direct method of frequency response analysis but in terms of generalized coordinates. The number of generalized coordinates typically used in the modal solution method is significantly lower than the number of physical coordinates, so direct integration of equations of motions is not as computationally intensive as in

the case of the direct method of frequency response analysis. However, if you specify damping for each normal mode, you can use the mode division method because in this case the independence condition of normal vibration modes is preserved, therefore, the equations remain uncoupled. This approach with damping introduced is known as specifying modal damping, when every i -th normal mode has corresponding damping b_i . The equation of motion for the i -th normal mode is of the form:

$$-\omega^2 m_i \xi_i(\omega) + i\omega b_i \xi_i(\omega) + k_i \xi_i(\omega) = p_i(\omega) \quad (3.40)$$

Solving the equation for each normal mode yields the response of the system:

$$\xi_i(\omega) = \frac{p_i}{-m_i \omega^2 + i b_i \omega + k_i} \quad (3.41)$$

The reverse transition from generalized coordinates to physical coordinates is described by equation (3.39) in a way similar to a system without damping.

The direct integration method gives a precise result but the computation efficiency rapidly falls off as the dimensionality of the problem increases. The modal method provides an approximate dynamic analysis solution. If all eigenvectors of the system are taken into account, the solution is precise and equal to the result obtained by direct integration. However, the experience of applying the modal method in practice shows that a correct approximation of the system's response can be achieved by taking into account only a small number of its first normal modes. It is recommended to include all normal modes of the system that fall in a the range that overlaps the range of excitation frequencies twofold or more.

Table 3.1 lists general recommendations for choosing the frequency response analysis method.

	Direct method	Modal method
Model dimensionality: – low	X	
		X
Number of excitation frequencies		

- few	X	
- many		X
High-frequency excitation		X
Accuracy		X

Example 3. Frequency response analysis using mode superposition

Launch NX and open the *Tower_sim1.sim* file. You automatically switch to NX Advanced Simulation.

Reset all dialog box settings to default by choosing the following main menu command: **Preferences→User Interface→General→Reset Dialog Box Settings**, and clicking the **OK** button.

1. Creating a new solution for modal frequency response analysis

In the model tree of the Simulation Navigator tab, right-click *Tower_sim1.sim* and choose *New Solution*. In the *Solution* dialog box (Figure 3.26), in the *Solution Type* list, select **SEDFREQ 111**. Change the name of the solution to *Modal_frequency_response*. The *Solution* dialog box stays open.

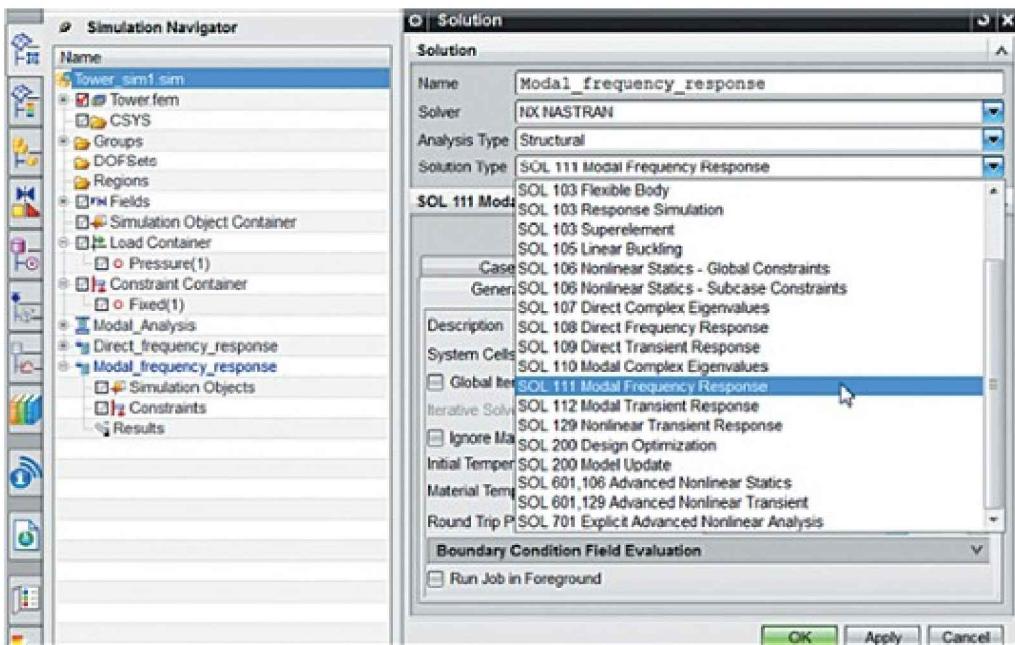


Figure 3.26. Creating a new solution

2. Specifying result output options

In the *Solution* dialog box select the *Case Control* tab and in the *Output Requests* list, select the **Structural Output Requests2** modelling object you created in the previous example.

3. Selecting the eigenvalue calculation method and specifying solution parameters

In the *Solution* dialog box, select the *Case Control* tab and specify the following:

- In the *Eigenvalue Method* – list, make sure *Lanczos* method is selected.
- Under *Lanczos Data*, click **Create Modeling Object**. In the *Real Eigenvalue – Lanczos2* dialog box (Figure 3.27), in the *Frequency Range – Upper Limit* box, enter 20 Hz, then click **OK**. In the *Solution* dialog box, click **OK** to open the *Solution Step* dialog box.

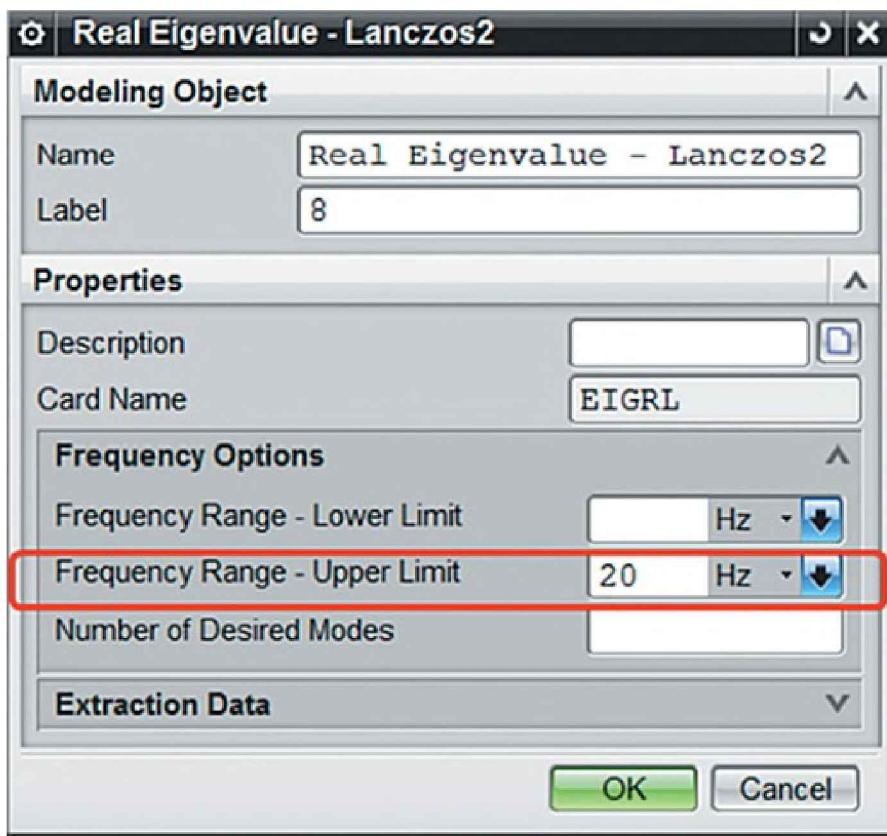


Figure 3.27. Specifying parameters for finding eigenvalues

4. Specifying excitation frequencies

In the **Solution Step** dialog box, change the name to *Modal Frequency*. In *Forcing Frequencies*, click **Create Forcing Frequencies**. In the **Modeling Object Manager** dialog box, click **Create**. In the **Forcing Frequencies – Modal1** dialog box (Figure 3.28), specify the following:

- In the *Frequency List Form* list, select *FREQ1*.
- In the *First Frequency* box, enter 1 Hz.
- In the *Frequency Increment* box, enter 0.2 Hz.
- In the *Number of Frequency Increments* box, type 45.

Click **OK**.

In the **Modeling Object Manager** dialog box, click **Create** again. In the new **Forcing Frequencies – Modal2** dialog box (Figure 3.28), specify the following:

- In the *Frequency List Form* list, select *FREQ4*.
- In the *Lower Bound* box, enter 1 Hz.
- In the *Upper Bound* box, enter 10 Hz.
- In the *Frequency spread* box, enter 0.3.
- In the *Number of evenly spaced frequencies* box, type 10.

Click **OK**.

The **Modeling Object Manager** dialog box automatically opens. Do the following:

- Hold *Ctrl* and click the two created modelling objects to select them.
- Click *Add to list*.
- Click *Close*. The **Solution Step** dialog box appears.



Figure 3.28. Specifying excitation frequencies

5. Specifying damping

In the **Solution Step** dialog box (Figure 3.29), specify the following:

- In the *Damping type* list, select *Structural*.
- In the *Structural damping* box, enter a value of 0.08, which is common to all frequencies.

Click OK.

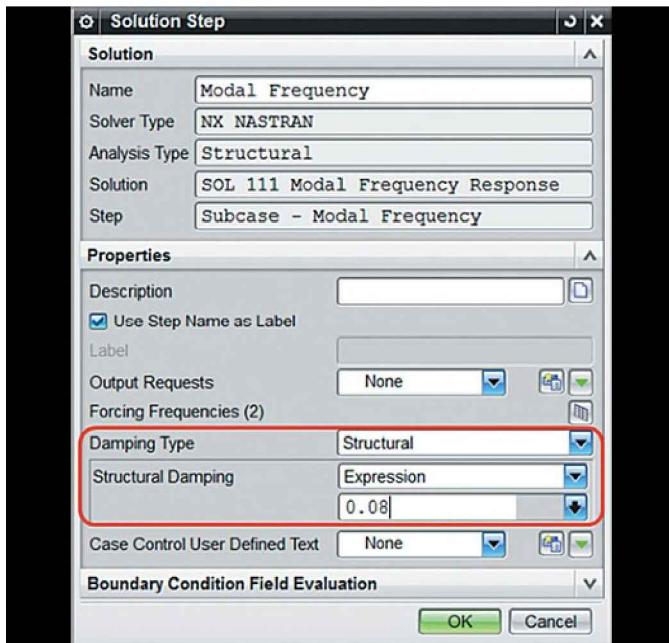


Figure 3.29. Specifying damping

Note that the model tree of the Simulation Navigator tab, has a new *Modal_frequency_response* node corresponding to the created solution.

6. Specifying loads and constraints

To specify boundary constraints, in the model tree of the Simulation Navigator tab, expand the *Constraint container* node and drag the previously created *Fixed(1)* constraint into the *Constraints* node of the *Modal_frequency_response* solution (Figure 3.30). In a similar way, drag the *Pressure (1)* load into the *Loads* node of the *Modal_frequency_response* solution.

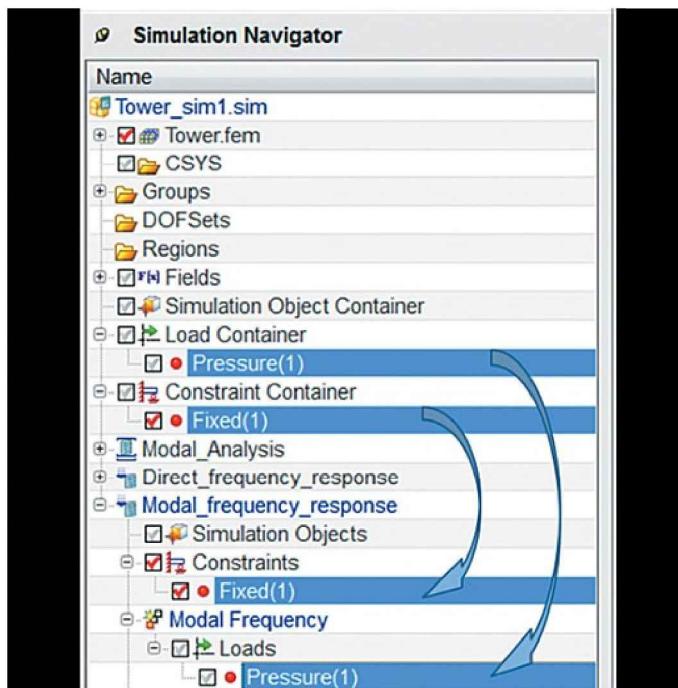


Figure 3.30. Specifying boundary constraints

The simulation model is ready for calculation. To save the model, choose **File→Save** in the main menu.

7. Running the solution

To run the solution, right-click *Direct_frequency_response* and choose **Solve...** in the shortcut menu. In the **Solution dialog box**, click **OK**. The computation can take several minutes. When the simulation finishes, close all new dialog boxes.

8. Review the results

Go to the **XY Function Navigator** tab of the Resource Bar. Right-click the *Nastran OP2 Files-Sort2 Results* node and choose **Open** in the shortcut

menu. Specify the *tower_sim1_modal_frequency_response.op2* results file. When the file opens, in the **XY Function Navigator** tab, a new *tower_sim1_modal_frequency_response* node appears. Expand all nodes, then hold *Ctrl* and click Node 4370: *Ty* components to select them, then right-click and choose the **Plot XY** command in the shortcut menu. The graphics area shows two frequency response graphs for the two methods (Figure 3.31). Compare the results of the direct and modal frequency response analyses. Use commands of the **XY Graph** toolbar to edit the graph as necessary (refer to Chapter 5, Part 1).

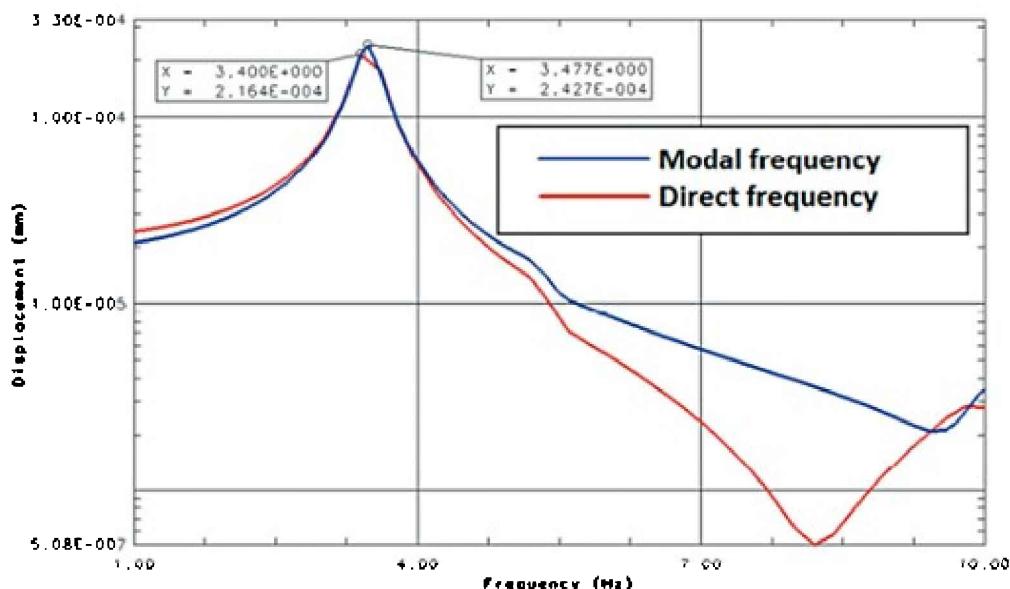


Figure 3.31. Direct and modal frequency response analysis results

3.5. Transient analysis

Transient analysis involves analysing dynamic behaviour of a system undergoing unsteady-state vibration. In this case the external load is a function of time that is defined at each point of the time span when it is active.

Similarly to frequency response analysis, transient analysis in NX Nastran can be carried out using two methods. The first is direct integration of equations of motion with the *SETRAN 109* solution (*Direct Transient Response* analysis). The second is the modal method (vibration mode superposition method), which uses the *SEMTRAN 112* solution (*Modal*

Transient Response analysis). The choice of the method depends on the size of the system being studied, the required result precision, the need to analyse the high-frequency region, and other considerations.

3.5.1. Direct method

The main feature that distinguishes transient analysis by direct integration of equations of motion (3.17) from frequency response analysis is that the response of the structure is found at discrete points of the time span in accordance with the time integration step value Δt . In other words, to carry out direct transient analysis, you need to solve a system of linear differential equations of the second order.

$$[M]\{\ddot{u}(t)\} + [B]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\} \quad (3.42)$$

The solution of each subsequent point in the time range $t + \Delta t$ is based on the solution for the previous step t . To achieve this, physical coordinates are represented by finite central differences [7]:

$$\{\dot{u}_n\} = \frac{1}{2\Delta t} \{u_{n+1} - u_{n-1}\}$$

$$\{\ddot{u}_n\} = \frac{1}{\Delta t^2} \{u_{n+1} - 2u_n + u_{n-1}\} \quad (3.43)$$

After substituting (3.43) and transforming the equation of motion, it takes the form:

$$[A_1]\{u_{n+1}\} = [A_2] + [A_3]\{u_n\} + [A_4]\{u_{n-1}\} \quad (3.44)$$

where $[A_1] = \left[\frac{M}{\Delta t^2} + \frac{B}{2\Delta t} + \frac{K}{3} \right]$ is the dynamic matrix.

$[A_2] = \frac{1}{3}\{P_{n+1} + P_n + P_{n-1}\}$ $[A_2] = \frac{1}{3}\{P_{n+1} + P_n + P_{n-1}\}$ is the load matrix averaged over three neighbouring points in time.

$$[A_3] = \left[\frac{2M}{\Delta t^2} - \frac{K}{3} \right] \text{ and}$$

$[A_4] = \left[-\frac{M}{\Delta t^2} + \frac{B}{2\Delta t} - \frac{K}{3} \right]$ are matrices that characterize the previous step solution.

Note that this book discusses linear analysis of transient processes, so matrices $[M]$, $[B]$, $[K]$ are assumed to be constant in time. The transient analysis solution is obtained by decomposing matrix $[A_4]$. If the integration step Δt is constant throughout the solution, matrix $[A_4]$ is decomposed once. At each subsequent time increment, only a new load vector has to be substituted, therefore, computation intensity of problems of high dimensionality is significantly alleviated. Obviously, the total computation time is proportional to the number of integration steps.

The time integration step needs to be large enough to correctly describe the variability of load in time and the behaviour of the system at the maximum natural frequency of interest. Using at least ten solution steps per response period at the natural frequency of interest is recommended.

As previously noted, you cannot explicitly specify structural damping in transient analysis, so it has to be expressed as equivalent viscous damping. The damping matrix $[B]$ includes several components:

$$[B] = [B^1] + [B^2] + \frac{G}{W^3} [K] + \frac{1}{W_4} \sum G_E [K_E] \quad (3.45)$$

where $[B]$ is the matrix which characterizes viscous damping of some finite element types: *CVISC*, *CDAMPi*, *CBUSH*, *CELASI*.

G is the total structural damping coefficient as specified by the **PARAM,G** parameter.

W_3 is the structural damping to equivalent viscous damping transformation frequency specified by the **PARAM,W3,r** parameter.

$[K]$ is the global stiffness matrix.

$[K_E]$ is the elemental stiffness matrix.

G_E is the elemental structural damping coefficient that is specified in material properties (**GE**).

W_4 is the elemental structural damping to equivalent viscous damping transformation frequency specified by the **PARAM,W4,r** parameter.

The first natural vibration mode of the structure is typically used as the transformation frequency [7].

Example 4. Analyzing transient processes using the direct method

To illustrate transient process analysis, it is useful to consider the dynamic behavior of a tower experiencing an impact.

Launch NX and open the *Tower_sim1.sim* file. You automatically switch to NX Advanced Simulation.

1. Creating a new solution for direct transient analysis

In the model tree of the Simulation Navigator tab, right-click *Tower_sim1.sim* and choose *New Solution*. In the *Solution* (Figure 3.32) dialog box, in the *Solution Type* list, select *SEDRAN 109*. Change the name of the solution to *Direct_transient_response*. The *Solution* dialog box stays open.

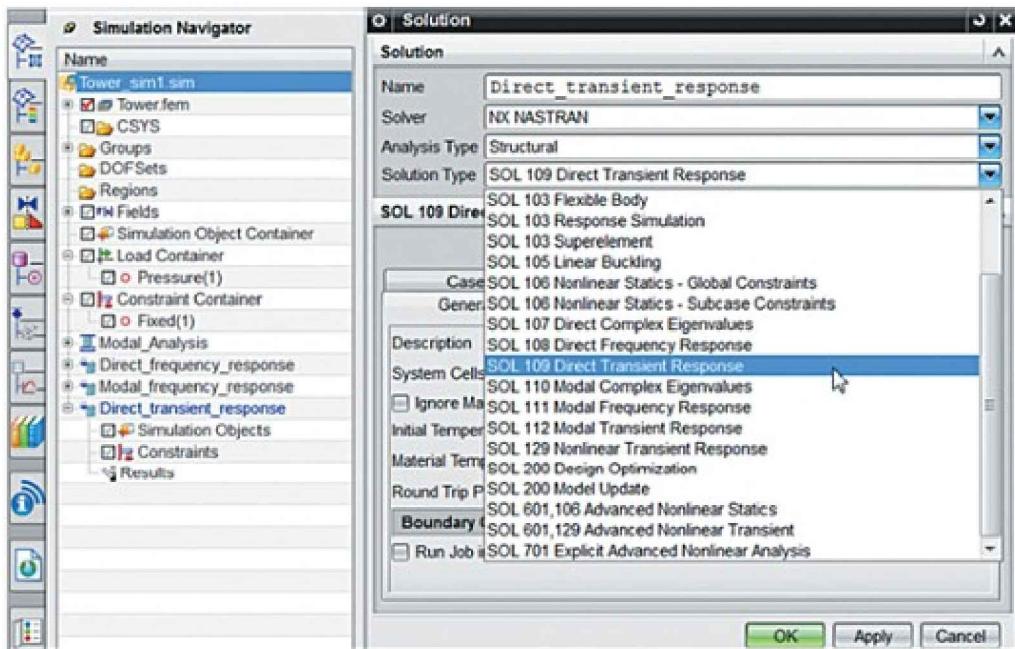


Figure 3.32. Creating a new solution

2. Specifying result output options

In the *Solution* dialog box, select the *Case Control* tab and next to the *Output Requests* option, click **Create Modeling Object**. In the *Structural Output Requests* dialog box (Figure 3.33), click *Disable All* and set the following options:

- In the *Stress* tab, do the following:

- a. Select the *Enable STRESS Request* check box.
- b. In the *Sorting* list, select *SORT2*.
- In the *Displacement* tab:
 - a. Select the *Enable DISPLACEMENT Request* check box.
 - b. In the *Sorting* list, select *SORT2*.
 - c. Click *OK* in the **Structural Output Requests3** dialog box.

The **Solution** dialog box stays open.

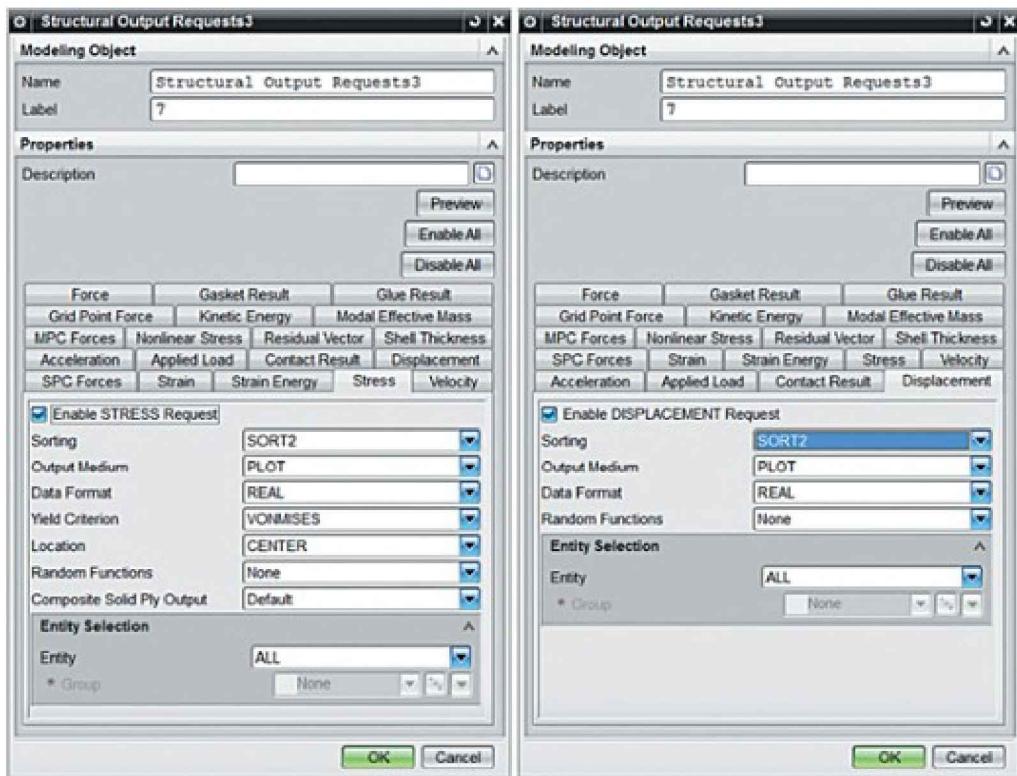


Figure 3.33. Specifying results output options

Note that in the model tree of the **Simulation Navigator** tab, there is a new *Direct_transient_response* node corresponding to the created solution.

3. Specifying the structural damping coefficient G

In the **Solution** dialog box, select the (*Parameters* tab and specify the following (Figure 3.34):

- In the *Damping* box, enter the structural damping coefficient value of 0.08.
- In the *Dominant frequency* dialog box, specify the value of the first

natural frequency of 3.47 Hz.

Click **OK**.

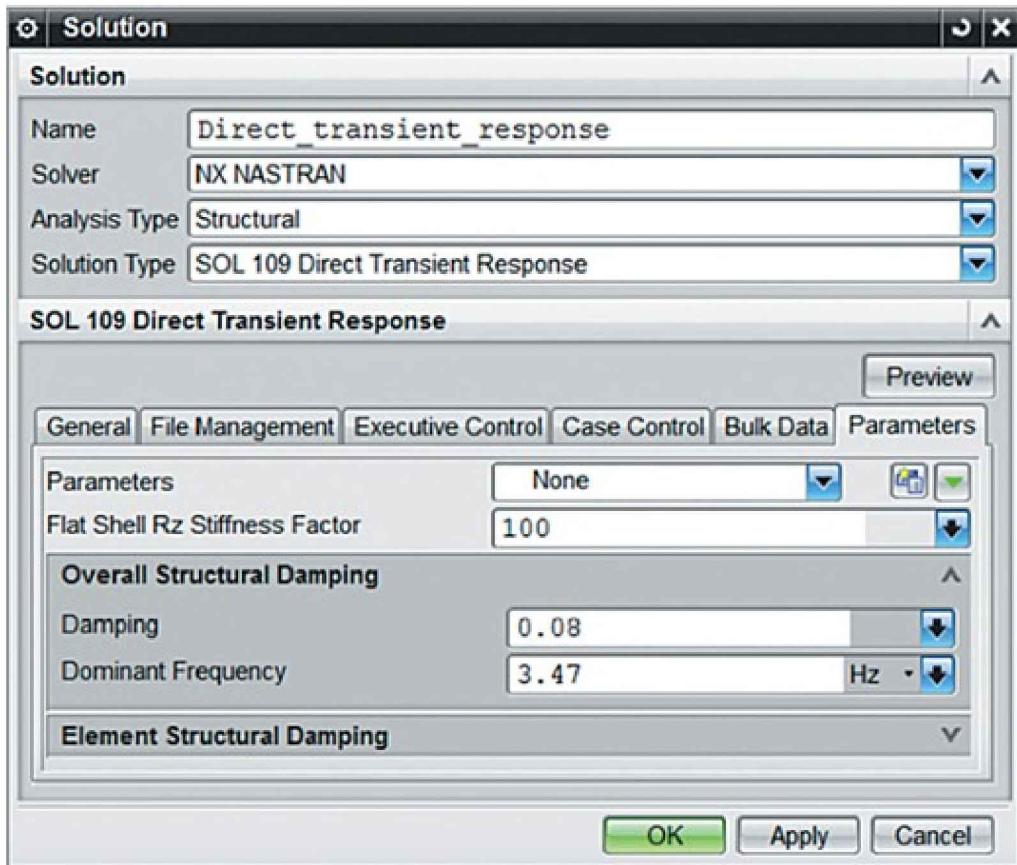


Figure 3.34. Specifying damping

4. Specifying the time integration step

To specify the time integration step, in the model tree of the Simulation Navigator tab, right-click the *Subcase – Direct Transient 1* node that corresponds to the step of the transient analysis solution, then choose *Edit*. In the *Solution Step* dialog box, do the following:

- Change the name to *Direct Transient*.
- In the *Time step definition* list, make sure *Single interval* is selected.
- Under *Time step interval*, click **Create modeling object**. In the *Time Step1* dialog box, specify the following (Figure 3.35):
 - a. In the *Number of time steps* box, type 200.

- b. In the *Time increment* box, enter 0.005 sec.
- c. In the *Skip factor for output* box, type 1.

Click **OK** in all dialog boxes.



Figure 3.35. Specifying the time integration step

5. Specifying loads and boundary constraints

5.1. To specify degree of freedom constraint conditions for the structure, in the model tree of the Simulation Navigator tab, right-click *Constraint container* to display the shortcut menu. In the submenu, choose the *User Defined Constraint* command. In the dialog box, specify the following (Figure 3.36):

- Under *Select Object*, select node 20000 (independent node of RBE2 elements).
- In the *DOF2* list, select *Free*. For all other degrees of freedom, select *Fixed*.

Click **OK**.

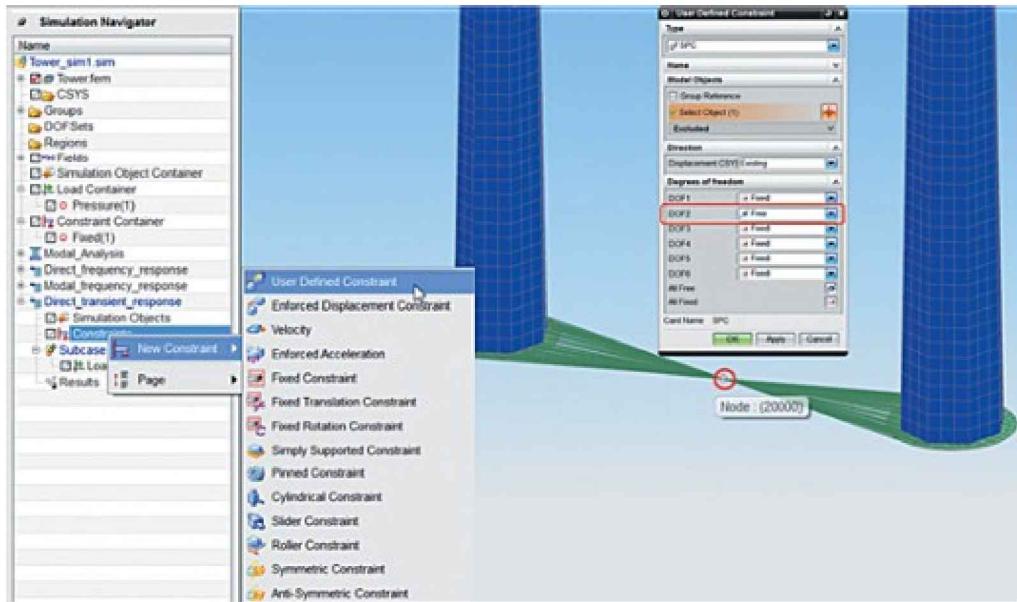


Figure 3.36. Specifying a *UserDefined(1)* constraint

The *UserDefined(1)* constraint is saved in the *Constraints* node of the *Direct_Transient_Response* active solution.

5.2. The external influence in this example is the time dependence of acceleration shown in Figure 3.37. The corresponding tabular values are saved in the *Shock_spectrum.csv* file.

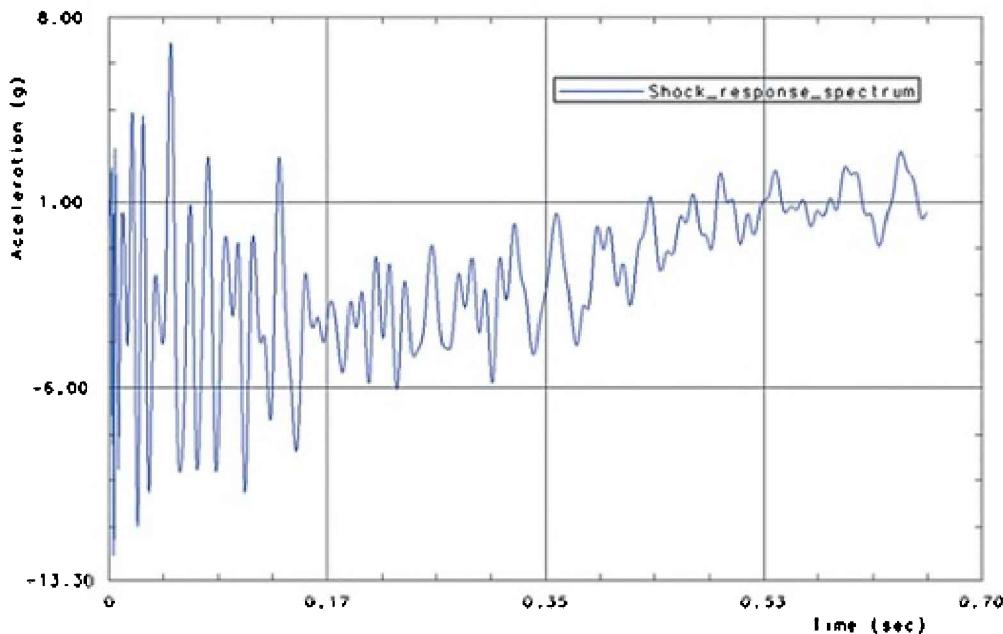


Figure 3.37. Impact

In the Simulation Navigator tab, right-click *Constraint container* to open the shortcut menu. In the submenu, choose the Acceleration command. In the dialog box, specify the following (Figure 3.38):

- Under *Select Object*, select node 20000 (independent node of RBE2 elements).
- In the *DOF2* list, select *Field*.
- Under *Define Field*, select *Table Constructor*. In the *Table Field* dialog box, specify the following:
 - In the *Domain* list of the *Independent* tab, select *Time*.
 - In the *Data Points* box, use the *Import from File* command and specify the *Shock_spectrum.csv* file in the working folder.

Click **OK**.

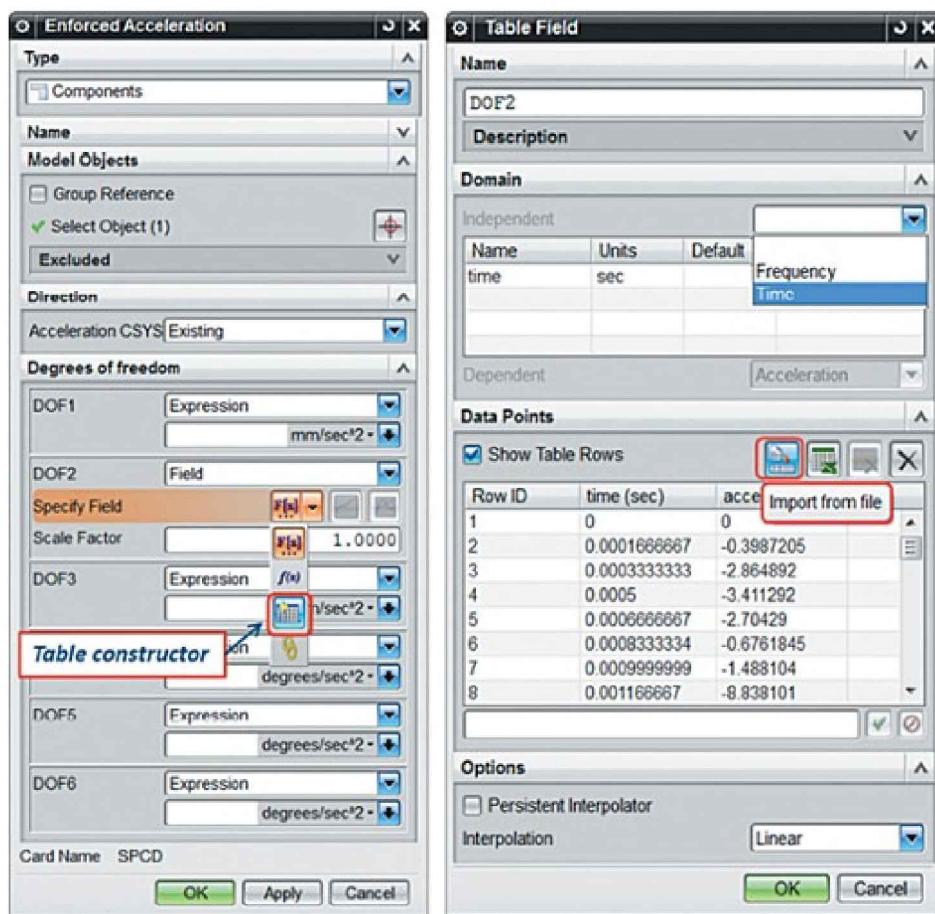


Figure 3.38. Specifying the time dependence of acceleration

The simulation model is ready for calculation. To save the model, choose **File→Save** in the main menu.

6. Running the solution

To run the solution, right-click *Modal_Analysis* and choose **Solve...** in the shortcut menu. In the **Solution** dialog box, click **OK**. The computation can take several minutes. When the simulation finishes, close all new dialog boxes.

7. Viewing simulation results

7.1. Displaying the time dependence of stress

It is obvious that the largest stress in the tower structure is encountered at its base, so it is useful to display a time dependence graph for the stress of element 8347.

Go to the **XY Function Navigator** tab of the Resource Bar. Right-click the *Nastran OP2 Files-Sort2 Results* node and choose **Open**. Specify the *tower_sim1_direct_transient_response.op2* results file. With the file opened in the **XY Function Navigator** tab, expand the *tower_sim1_direct_transient_response* node, and then expand the *Stress-TOP (Subcase 1)* node. Double-click *Element 8347: Node 9279: XX* to plot the way the stress tensor component changes in time (Figure 3.39). Use commands of the **XY Graph** toolbar to edit the graph as necessary (refer to Chapter 5, Part 1).

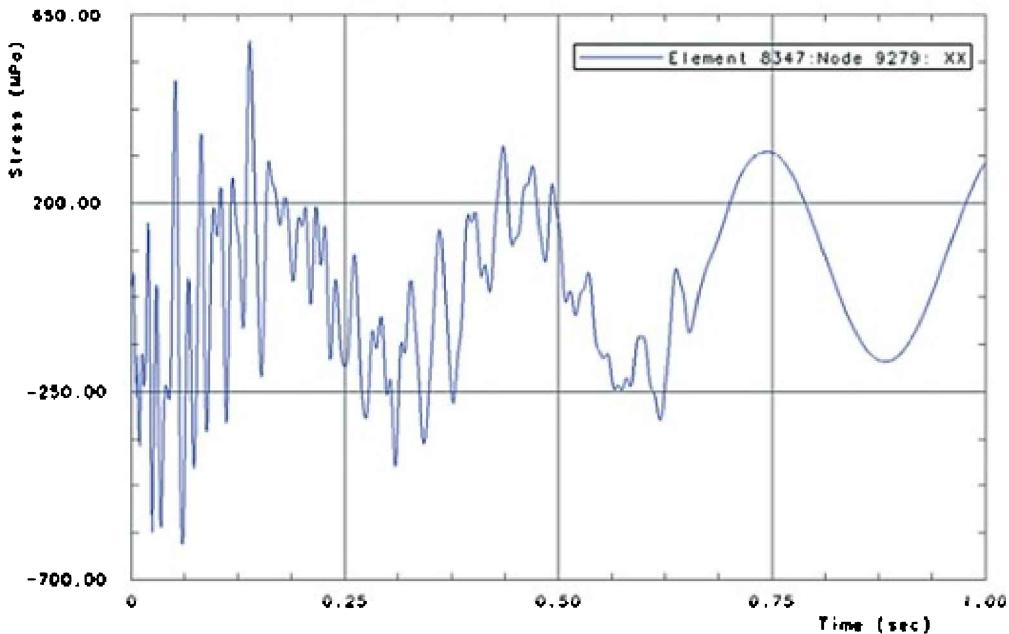


Figure 3.39. Time dependence of the XX stress tensor component in node 9279

7.2. Animating results

Select the **Post Processing Navigator** tab with the results loaded. To animate transient analysis results, do the following (Figure 3.40):

- Expand the *Direct_transient_response* node.
- Expand any node, for example, *Increment 161*.
- Expand the *Stress – Element-Nodal* node and double-click *Von Mises*.

On the **Post Processing** toolbar, click the **Animation** command. Set the following options in the dialog box (Figure 3.40):

- In the *Animate* list, select *Iteration*.
- Select the *Full-cycle check box*.

Click **OK**.

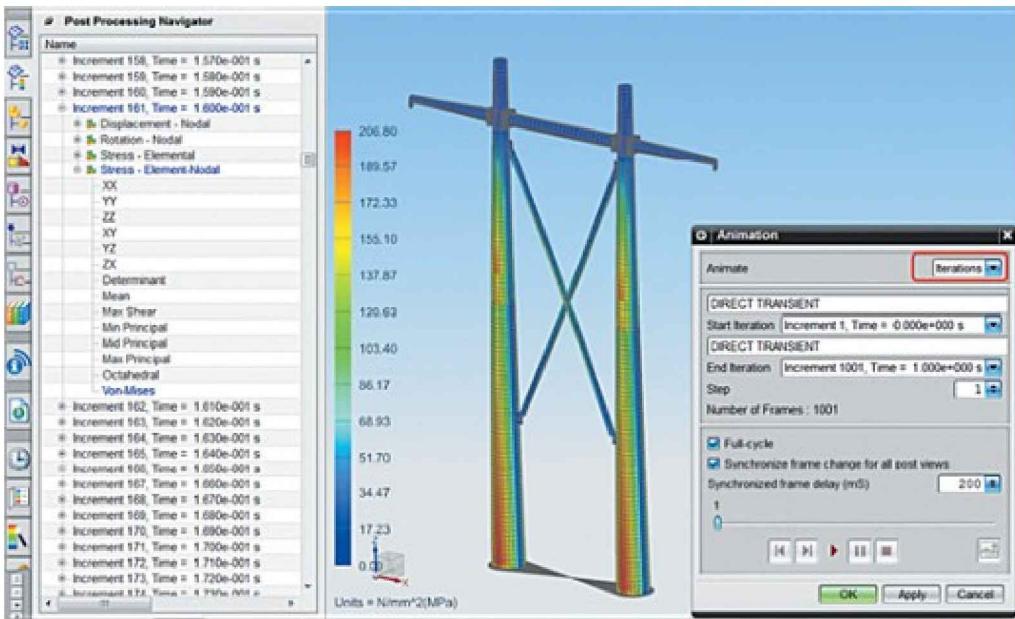


Figure 3.40. Viewing simulation results as an animation

3.5.2. Modal method

In large problems where there is a need to conserve computation resources, a modal method of transient process analysis is often used. This method is based on going from physical coordinates $\{u(\omega)\}$ to modal coordinates $\{\xi(\omega)\}$ in accordance with equation (3.36). The nature of the approach is detailed in para. 3.4.2.

When you use the modal method of transient analysis, the damping matrix $[B]$ and the global stiffness matrix are not diagonal, therefore, the independence condition of individual modal equations of motion is not satisfied. This difficulty is resolved by using the previously described modal damping approach. The equation of motion for the i -th normal mode is of the form:

$$m_i \ddot{\xi}_i(t) + b_i \dot{\xi}_i(t) + k_i \xi_i(t) = p_i(t) \quad (3.46)$$

In accordance with equation (3.39), the total reaction of the system is a superposition of its reactions for all relevant normal modes.

Table 3.2 lists general recommendations for selecting the direct method or the modal method of transient analysis.

	Direct method	Modal method
--	---------------	--------------

Model dimensionality:	X	
– low		X
– high		
Number of time integration steps:		
– few	X	
– many		X
High-frequency excitation		X
Accuracy		X
Initial Conditions		X

Example 5. Analysing transient processes using the modal method

Launch NX and open the *Tower_sim1.sim* file. You automatically switch to **NX Advanced Simulation**.

1. Creating a new solution for modal transient analysis

In the model tree of the Simulation Navigator tab, right-click *Tower_sim1.sim* and choose *New Solution*.

In the **Solution** (Figure 3.39) dialog box, in the **Solution Type** list, select **SEMTRAN 112**. Change the name of the solution to **Modal_transient_response**. The **Solution** dialog box stays open.

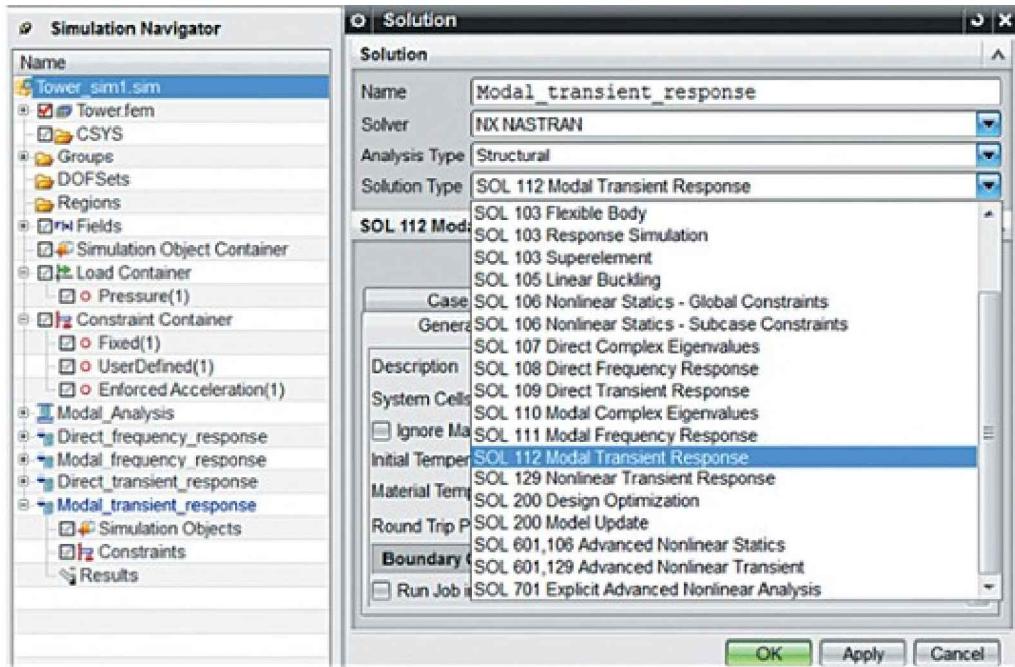


Figure 3.41. Creating a new solution

2. Specifying result output options

In the *Solution* dialog box, select the *Case Control* tab and in the *Output Requests* list, select the ***Structural Output Requests3*** modelling object you created in the previous example.

3. Choosing the natural frequency calculation method

In the *Solution* dialog box, select the *Case Control* tab and specify the following:

- In the *Eigenvalue Method* – list, make sure *Lanczos* method is selected.
- Under *Lanczos Data*, click ***Create Modeling Object***. In the ***Real Eigenvalue – Lanczos3*** dialog box, in the *Frequency Range – Upper Limit* box, enter 400 Hz. Keep default values for the other options. Click ***OK***.

In the *Solution* dialog box, click ***OK*** to open the *Solution Step* dialog box.

4. Specifying the time integration step and damping parameters of the system

In the *Solution Step* dialog box, specify the following:

- Change the name to *Modal Transient*.
- In the *Time step interval* list, select the *Time Step1* modeling object you created in the previous example.
- In the *Damping type* list, select *Structural*.
- In the *Structural Damping* box, enter 0.08.

Click **OK**.

5. Specifying loads and constraints

To specify boundary constraints, select the Simulation Navigator tab and expand the *Constraint container* node of the model tree. Drag the previously created *Userdefined(1)* and *Acceleration(1)* boundary conditions into the *Constraints* node of the *Modal_transient_response* solution (Figure 3.42).

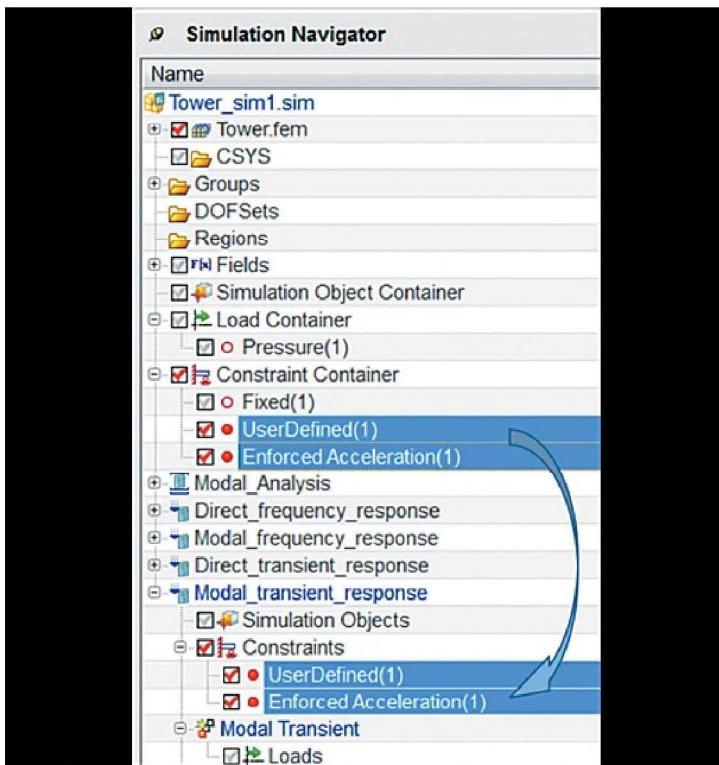


Figure 3.42. Specifying loads and boundary constraints

The simulation model is ready for calculation. To save the model, choose **File→Save** in the main menu.

6. Running the solution

To run the solution, right-click *Modal_frequency_response* and choose

Solve... in the shortcut menu. In the **Solution** dialog box, click **OK**. The computation can take several minutes. When the simulation finishes, close all new dialog boxes.

7. Review the results

Go to the **XY Function Navigator** tab of the Resource Bar. Right-click the **Nastran OP2 Files-Sort2 Results** node and choose **Open**. Specify the **tower_sim1_modal_transient_response.op2** results file. When the file opens, in the **XY Function Navigator** tab, a new **tower_sim1_modal_transient_response** node appears. Expand both nodes, hold **Ctrl** and click **Element 8347: Node 9279: XX** components to select them, then right-click and choose the **Plot XY** command in the shortcut menu. Compare the results of the direct and modal transient analyses (Figure 3.43). Use commands of the **XY Graph** toolbar to edit the graph as necessary (refer to Chapter 5, Part 1).

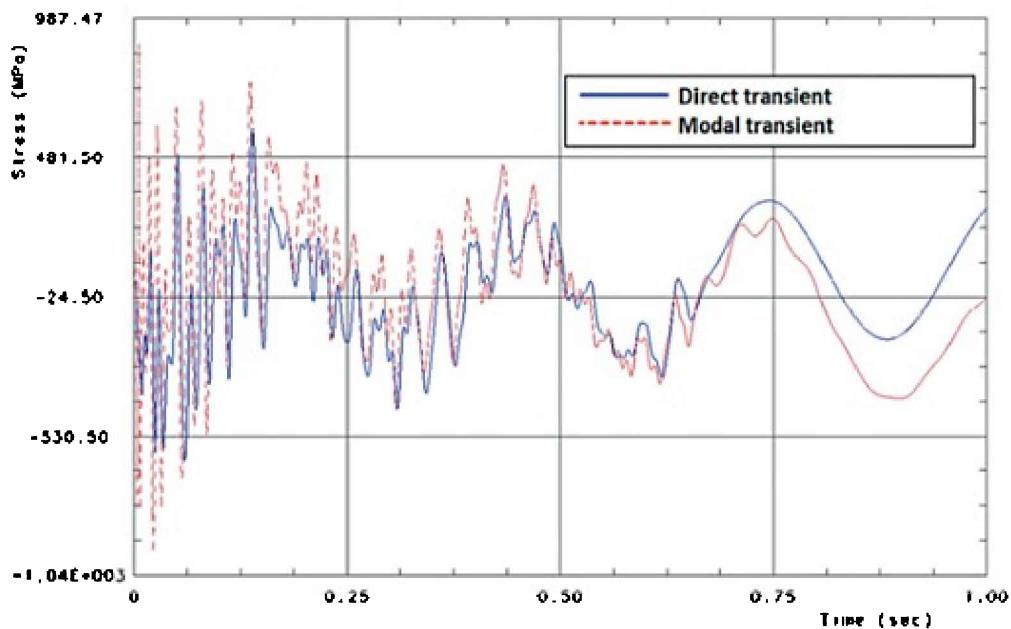


Figure 3.43. Direct and modal transient analysis results

3.6. Extra functionality for solving dynamic problems

3.6.1. Nonlinear dynamic analysis

This chapter describes the basic types of dynamic analysis and methods of solving dynamic problems in linear elastic formulation, which implies

constant mass, stiffness, and damping matrices with external loads varying with time. In many situations, it is acceptable to assume small displacements and strain when considering linear behaviour of structures. However, in some cases linear properties of the system are not enough, and nonlinear methods have to be used. Note that solutions of nonlinear dynamic problems are only applicable to unsteady-state vibration analysis (transient processes).

Nonlinear properties of structures can be divided into three categories:

- Nonlinearity of material properties.
- Geometrical nonlinearity, including large displacements, large strains and variation of the load vector as a function of displacement (follower load effect).
- Contact interaction of structures depending on the deformed state.

If nonlinear material properties or geometrical nonlinearity (large displacements) need to be taken into account, it is recommended to use NX Nastran solutions *NLTRAN 129* (nonlinear dynamic analysis of transient processes) and *NLSTATIC 106* (nonlinear static analysis). For solving dynamic problems with contact interaction of parts of the structure and/or large strains, you should use NX Nastran solutions *ADVNL 601*, *129* (advanced nonlinear dynamic analysis of transient processes) and *ADVNL 701* (explicit advanced nonlinear dynamic analysis).

In addition to the specified nonlinearity types, many systems have nonlinear elastic structural elements with inherently nonlinear force-displacement relation. Nonlinear elastic elements are common in many types of structures and mechanical systems. Here are some examples: spring cushions for piping systems, chain couplings (stiff only in tension), special devices for maintaining constant force, nonlinear shock protection systems, models of soils and foundations (stiff only in compression), and so on. [7]. There are two ways of solving such problems in NX Nastran. Using the general approach involves specifying nonlinear elastic properties of materials and/or using spring-damper finite elements such as *CBUSH*. Obviously, in comparison with linear dynamic analysis, this approach consumes a lot of computational resources for constructing the stiffness matrix of the nonlinear elastic element, solving equilibrium equations at each iteration, and evaluating convergence of the solution.

NX Nastran implements an additional efficient pseudo force method that can solve problems of this type. If this method is used, the effect of local (localized) nonlinearities is conceptualized as an additional applied force in the right-hand part of the equation of motion (3.17), which in turn allows you to avoid the added computation complexity. The nonlinear force is calculated at the current integration time step and used at the subsequent step. The equation of motion in this case is of the following form:

$$[M]\{\ddot{u}(t)\} + [C]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\} + \{N(t)\} + \{N(t - \Delta t)\} \quad (3.47)$$

where $\{N(t)\}$ is the added nonlinear force.

Note that the nonlinear force lags one step behind the current solution, requiring finer integration steps than for the linear solution.

3.6.2. Dynamic reduction

The state of the art allows millions of degrees of freedom in a FE analysis simulation model of a structure. Such fine detail is needed to solve static stress problems. In most cases of dynamic analysis, this level of detail with the consequent computation and time requirements is excessive. As discussed before, correct description of dynamic behaviour of structures only requires taking into account the lowest vibration frequencies within the working frequency band that is 1.5 or 2 times wider than the highest frequency of interest. The numerical approach, which allows the reduction of the simulation model to a model with significantly fewer degrees of freedom for dynamic analysis in NX Nastran, is called dynamic reduction. It is important to understand that reduction is an approximate method of solving dynamic problems. Its precision depends on the number of normal vibration modes that remain in the system after reduction. The most widespread method of solving dynamic problems is the dynamic synthesis method, also known as the Craig-Bampton method [10].

3.6.3. Using superelements

In essence, the use of superelements involves splitting the whole FE model into several parts (submodels), each of which then undergoes dynamic reduction to represent it with reduced mass, stiffness, and damping matrices. A superelement is thus a reduced representation of one of the parts of the FE model. Solutions are calculated separately for each

superelement, and all results are then combined to simulate the full FE model (Figure 3.44).

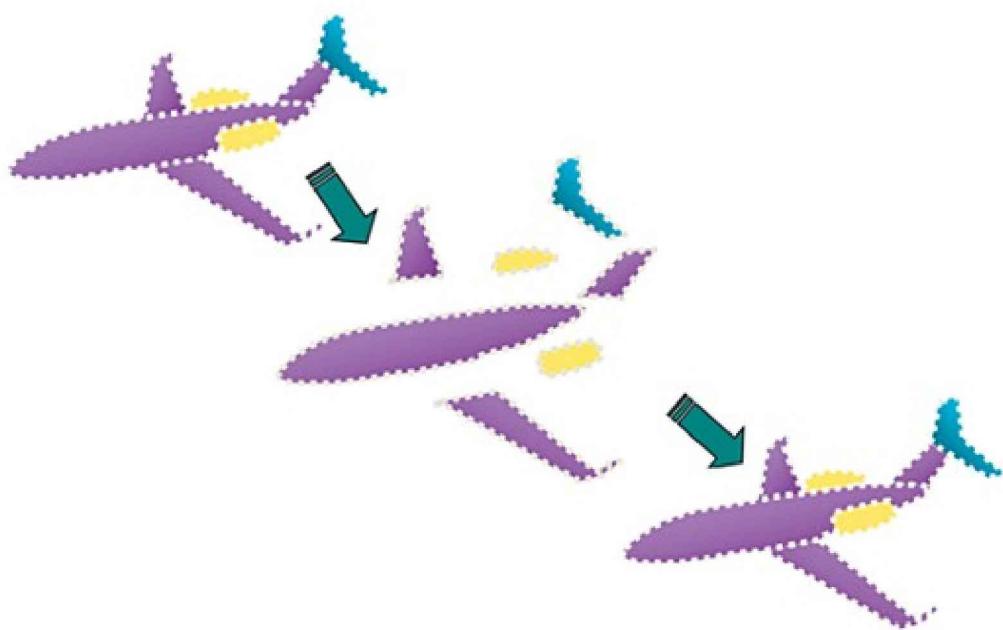


Figure 3.44. Schematic of the superelement approach

The dynamic synthesis method with superelements involves defining global mass, stiffness, damping matrices and load vectors in terms of generalized coordinates (normal modes of the system) and physical coordinates (degrees of freedom at boundaries of superelements).

The superelement method is advantageous in a number of ways:

- Ability to solve large problems of.
- Less computation power required.
- Shorter computation time.
- Convenient workflow.
- Sharing results and FE models without topology, that is, without compromising the confidentiality of the developed model.

Chapter 4. Nonlinear static and dynamic analysis

In practice, depending on the loading conditions, most structures exhibit nonlinear behaviour. In many situations, however, it is acceptable to assume linear behaviour with small displacements and strains. In other cases, correct results cannot be obtained without accounting for nonlinear behaviour.

With NX Nastran Advanced Nonlinear module, you can carry out nonlinear analysis with contact interaction, nonlinearity of material properties, and geometrical nonlinearity. The Advanced Nonlinear solver built into NX Nastran is the ADINA nonlinear solver of worldwide renown, which is implemented as the ADVNL SOL 601 solution for the implicit integration scheme and as ADVNL SOL 701 for the explicit integration scheme .

4.1. Introduction to nonlinear analysis. Special aspects of FE modelling

A structure's behaviour can be considered nonlinear in cases where loads, material properties, conditions of contact interaction, or the stiffness of the system are functions of displacements. The assumption of a linear relation between strain and displacement that underlies linear analysis does not hold true in the case of large displacements and rotations.

There are three types of nonlinearity:

- Geometrical nonlinearity, encountered where stiffness properties or loads change greatly as a result of deformation.
- Material nonlinearity, that accounts for the nonlinear behavior of materials.
- Nonlinearity of boundary conditions, namely, contact interaction between surfaces of bodies.

Compared to a linear analysis, nonlinear analysis requires more computing resources and usually a more qualified engineer. The same finite elements are used for linear and nonlinear analyses, therefore, you can quickly switch from a linear problem to a nonlinear one, and vice versa

NX Advanced Simulation provides the following NX Nastran solution types for nonlinear analysis (Figure 4.1):

- NLSTATIC SOL 106 (nonlinear static analysis).
- NLTRAN SOL 129 (nonlinear dynamic analysis).
- NLSCH SOL 159 (unsteady-state nonlinear thermal analysis).
- ADVNL SOL 601, 106 (advanced nonlinear static analysis).
- ADVNL SOL 601, 129 (advanced nonlinear dynamic analysis, implicit).
- ADVNL SOL 701, (advanced nonlinear dynamic analysis, explicit).

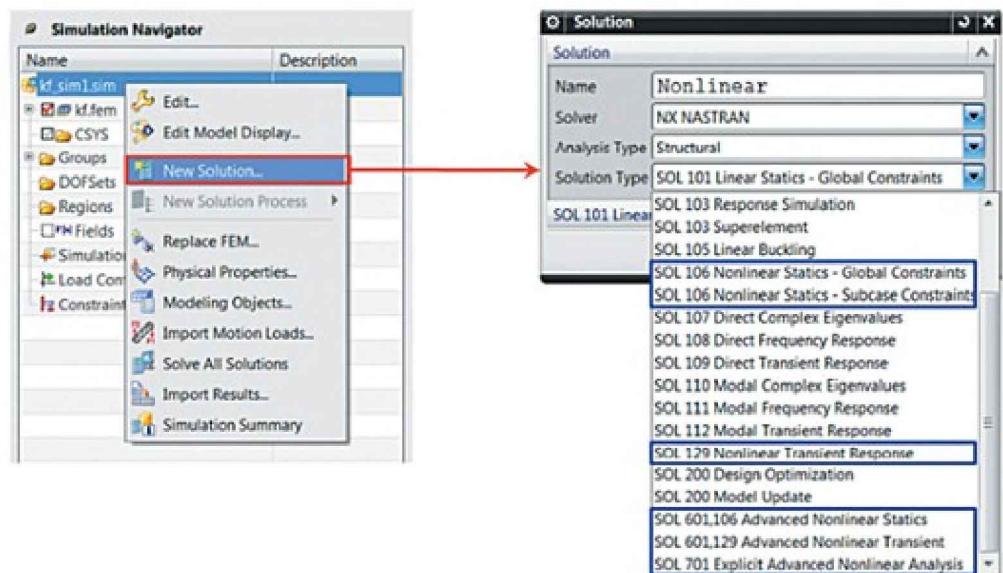


Figure 4.1. Types of NX NASTRAN nonlinear solutions

The following table lists the basic nonlinearity features for NLSTATIC 106/129, ADVNL 601,106/129, and ADVNL 701 solutions:

Feature	SOL 106/129 Basic nonlinear	SOL 601 Advanced nonlinear implicit	SOL 701 Advanced nonlinear explicit
Contact interaction	Gap elements	Surface, contact of edges, and gap elements	Surface
Large displacements	Yes	Yes	Yes

Large strains	No	Yes	Yes
Elastoplasticity	Yes	Yes	Yes
Hyperelastic material models	1	5	3
Material creep	Yes	Yes	No
Seal-type material model	No	Yes	No
Glue connection	Yes (except problems with large displacements)	Yes	No
Implicit solver	Yes	Yes	No
Explicit solver	No	No	Yes

You can solve problems with nonlinearities of various kinds using SOL 106/129, ADVNL 601,106/129 solutions. These solutions support all element types that are used in linear analysis with slightly different formulations. In the explicit ADVNL 701 solution, the set of usable FE elements is restricted as follows [23]:

- Plane stress and plane strain 2D elements are not supported.
- Laminate shells with *PCOMP* physical properties are not supported.
- Axisymmetric elements are not supported.
- Among shell elements, only *CQUAD4* and *CTRIA3* are supported.
- Among 3D elements, only *CHEXA8*, *CPENTA6*, and *CTETRA4* are supported.

You can use NX Nastran nonlinear solutions to apply constraints to all types of degrees of freedom and almost all load types, except the following:

- Angular centrifugal acceleration.
- Bolt pre-tensioning in ADVNL 701.
- Glue conditions in ADVNL 701.

Note that you can treat pressure as a follower load. This means: vary the load direction depending on the strained state of the model, only if large displacements are taken into account (*PARAM, LGDISP, 1 parameter*). The *LOADOPT* parameter allows the follower load effect with large

displacements to be neglected (Figure 4.2, A) since it causes an unsymmetric system matrix that is more difficult to solve.

Due to the peculiarities of solution methods used with nonlinear problems, the load should be applied gradually (over a few steps or substeps). This can be achieved by specifying a constant load value and an increment value in the static analysis of NLSTATIC 106 solution [6]. For the purposes of ADVNL 601,106 static analysis, the load can be constant (independent of time) or time-dependent. If the load is applied gradually to improve the convergence of the nonlinear solution, time acts as a load proportional factor. A notable exception is material creep in a static analysis, because material creep properties are time-dependent.

Dynamic analysis deal with the response of a structure to external influences that vary with time. It is important to choose an adequate time integration scheme and specify the initial conditions correctly.

Nonlinear analysis is applicable to various combinations of nonlinearity types. The most common scenario involves the need to account for elastoplastic material properties, large displacements, and contact interaction. Small displacements and strains are used by default. The *PARAM, LGDISP, 1* parameter specifies the treatment of large displacements, *PARAM, LGSTRN, 1* specifies the treatment of large strains. Note that whenever large strains are taken into account, also large displacements are considered (Figure 4.2, B). It is important to remember that large strains are taken into account only for 3D elements, 2D shell elements, axisymmetric, elements and plane straine elements.

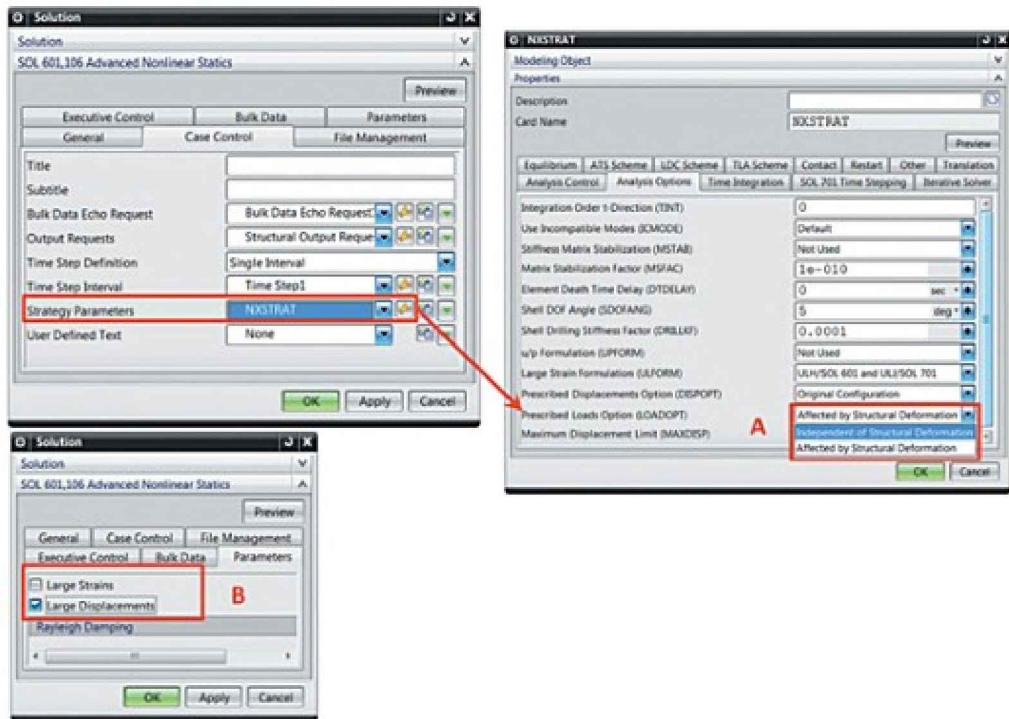


Figure 4.2. LOADOPT, LGDISP, and LGSTRN parameters

4.2. Geometrical nonlinearity

It is often necessary to consider geometrical nonlinearity effects when modelling the behaviour of structures with large displacements, large strains, follower loads, and post-buckling behaviour. Contact interaction is sometimes considered just another type of geometrical nonlinearity but in this chapter it is discussed separately. Geometric nonlinearity is relevant, for example, in the analysis of thin plates, rotation of shafts, metal molding, and so on. In other situations, the decision to account for geometrical nonlinearity is based on the engineer's experience. For example, if a plate is deflected by more than 20% of its thickness, or if a beam is deflected by more than 2% of its length, geometrical nonlinearity is typically taken into account (Figure 4.3).

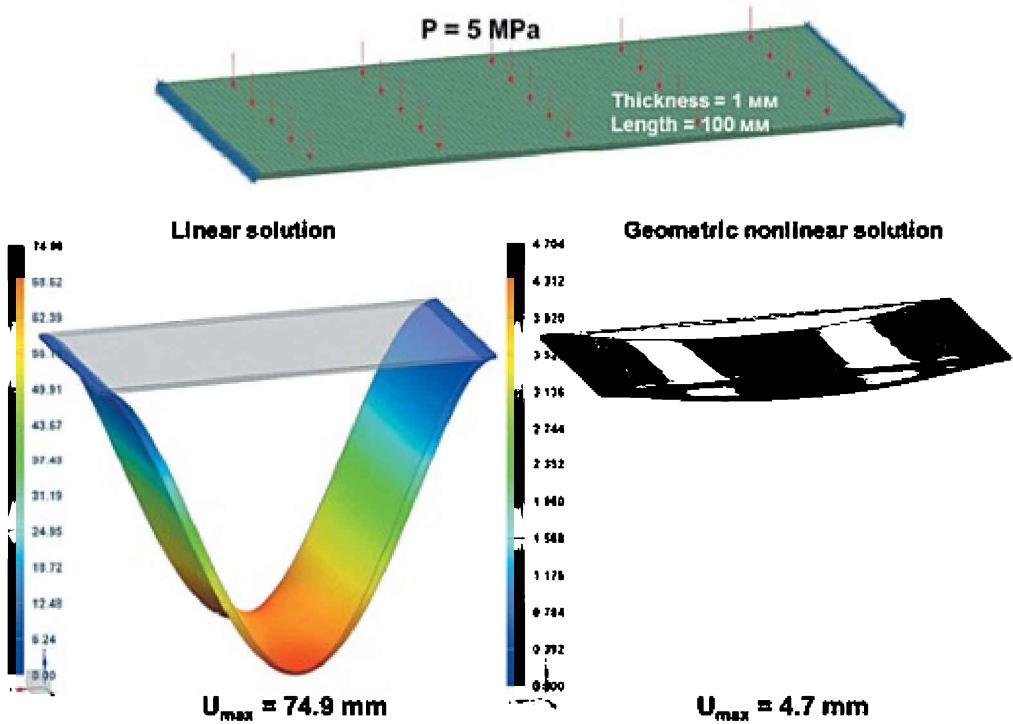


Figure 4.3. Accounting for geometrical nonlinearity in thin plate deflection

4.3. Nonlinearity of materials

In some loading conditions, nonlinear properties of materials have to be considered, for example, if hyperelastic materials are used, or if elastoplastic properties of metals are relevant. Nonlinear behaviour of materials is quite complex, therefore, depending on the situation and analysis type, established, well-known material models are used. These models describe the nonlinear effects by approximating the real behaviour.

4.3.1. Nonlinear material types

You can use the following material types in the SOL 106 solution [6]:

- Nonlinear elastic
- Hyperelastic
- Elastoplastic
- Viscoelastic
- Temperature-dependent

You can use the following combinations of materials and elements in

ADVNL 601 and 701 solutions [23]:

Material type/element type	Rod	Beam	Shell	2D Solid	3D Solid
Elastic isotropic	✓	✓	✓	✓	✓
... temperature-dependent	✓		✓	✓	✓
... with creep (only SOL 601)	✓		✓	✓	✓
Elastic orthotropic			✓	✓	✓
... temperature-dependent			✓	✓	✓
Plastic isotropic	✓ ¹	✓ ^{1, 2}	✓	✓	✓
... temperature-dependent	✓		✓	✓	✓
... with creep (only SOL 601)	✓		✓	✓	✓
Hyperelastic: Mooney-Rivlin, Ogden, Sussman-Bathe, Arruda-Boyce (only SOL 601), Hyperfoam (only SOL 601)				✓	✓
Viscoelastic with plasticity (only SOL 601)	✓		✓	✓	✓
Nonlinear elastic	✓		✓	✓	✓
Seal (only SOL 601)					✓
With shape memory (only SOL 601)	✓		✓	✓	✓

¹ Temperature strain is neglected. ² Bilinear plasticity model only.

Note that hyperelastic properties are implemented in *PLPLANE* physical properties for 2D elements and *PLSOLID* physical properties for 3D elements. Large strains are necessarily taken into account.

The following section provides a detailed description of the most commonly encountered use case for nonlinear materials, that is, working with elastoplastic properties. Detailed information about other nonlinear materials is provided in [23].

4.3.2. Elastoplastic material

When the linear static method is used, stress is assumed to be linearly proportional to strain. A linear relation of stress and strain is established using Young's modulus. If the stress in a structure exceeds the yield strength, then plastic properties of the material are typically taken into account.

Plasticity is the ability of a body to maintain a deformed state after loading is fully or partially lifted. Nonlinear analysis in NX Nastran allows nonlinear stress-strain relations to be taken into account and to calculate the residual plastic deformation that exists in the structure after the loading is removed.

NX Nastran supports the following plasticity criteria:

- von Mises: used for plastic materials in most cases, pressure independent.
- Tresca: used for brittle materials and some pressure independent plastic materials.
- Drucker-Prager: used for materials such as soil and concrete with internal friction so that yield depends on the hydrostatic pressure.
- Mohr-Coulomb: used for materials such as stone where the compressive strength exceeds the tensile strength.

In case of pressure independent plasticity, calculations beyond the elastic limit are mainly based on experimental studies of material properties in uniaxial tension and compression. Results of such studies are typically represented as stress-strain relation graphs. To simplify calculations beyond the elastic limit, elongations are replaced by functions (curves or straight lines) that have simple mathematical formulations but agree well with experimental results [24]. NX Nastran supports two methods of specifying material stress-strain diagrams:

- As a bilinear curve that is defined by the initial plastic stress value (*LIMIT1*) and the slope (*H*) of the straight-line section, which corresponds to plastic deformation (Figure 4.4, A).
- As a piecewise linear relation that is defined by tabular stress-strain values (Figure 4.4, B), where the first segment corresponds to elastic deformation.

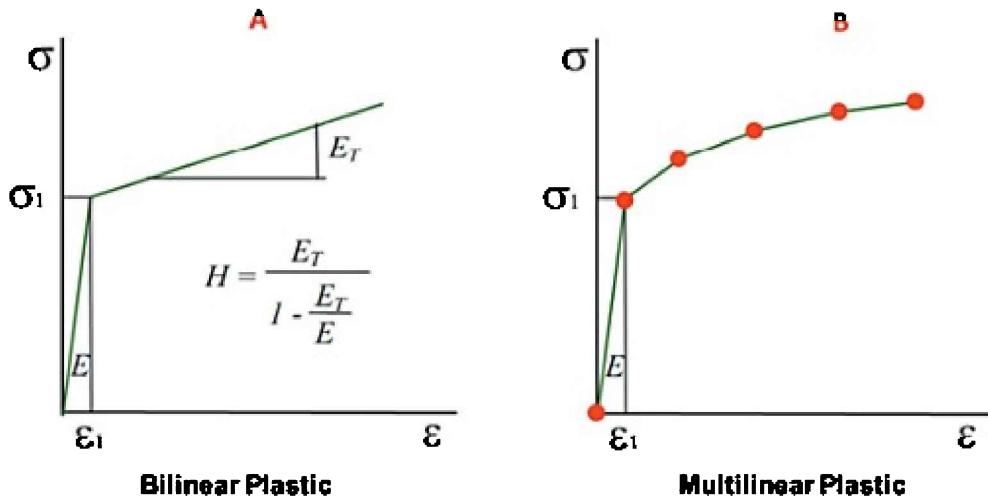


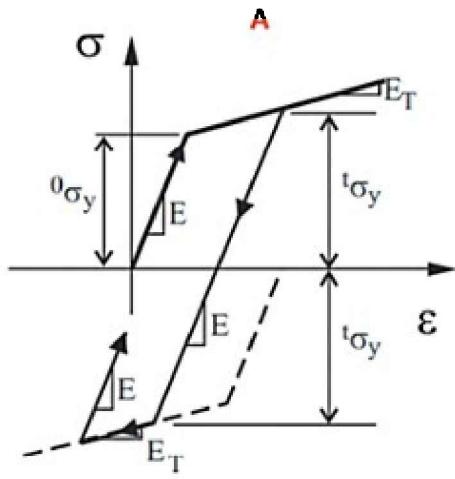
Figure 4.4. Specifying strain diagrams

In plastic deformation, the yield surface of the material can change with increasing strain. This phenomenon is called strain hardening. If a material was subjected to an axial force (for example, tension) in the plastic deformation zone, this material's ability to resist plastic deformation of the opposite sign (compression) is reduced. This phenomenon is called the Bauschinger effect and describes the relationship between isotropic and kinematic hardening.

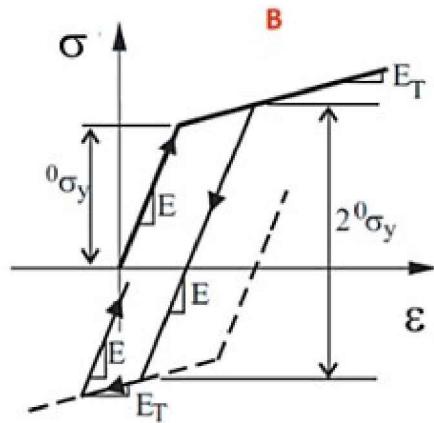
NX Nastran supports three hardening models:

- Isotropic hardening, in which the plasticity point under compression is equal to tensile plastic deformation (Figure 4.5, A) so that the yield surface opens equally.
- Kinematic hardening, in which the elastic unloading value is equal to twice the value of initial plastic stress (Figure 4.5, B) so that the yield surface moves in the stress space.
- Combined hardening, which represents the behaviour of the material as a combination of kinematic and isotropic hardening.

Thus, the yield criterion determines the initial yield surface. The choice of the hardening model defines the way it changes and the stress state upon repeated plastic deformation.



Bilinear isotropic hardening



Bilinear kinematic hardening

Figure 4.5. Bilinear hardening models

4.3.3. Specifying plastic properties in NX

The following rules must be followed when specifying plastic properties of materials in tabular form:

- Kinematic hardening cannot be used.
- The first point must be 0, 0.
- Initial plastic stress (*LIMIT1*) and Young's modulus must be specified.
- The second point must correspond to the specified *LIMIT1* value.
- The slope of the line connecting the first and the second point must be equal to Young's modulus.
- All other points are entered in terms of effective plastic deformation.
- The last point corresponds to the point of failure. You can use the *XTCURVE=0* parameter to “remove” elements whose strain values are larger than the values at the point of failure. The *DTDELAY* parameter delays the “removal” of these elements to improve convergence of the solution (Figure 4.6).

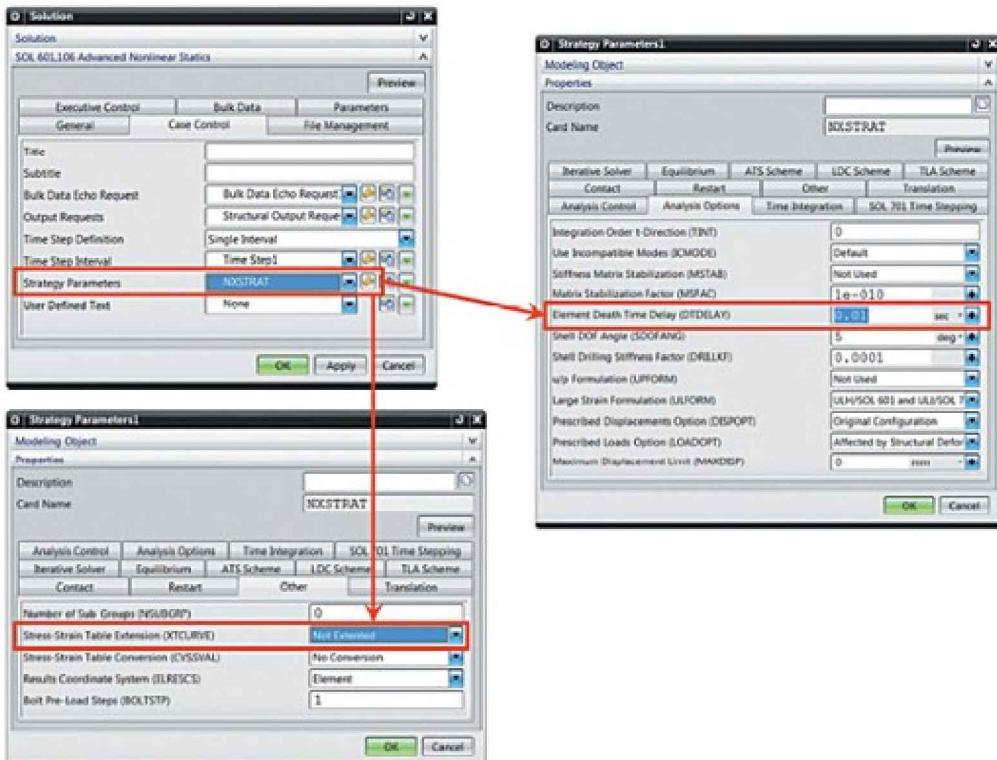


Figure 4.6. XTCURVE and DTDELAY parameters

To specify elastoplastic material properties in NX Advanced Simulation, do the following (Figure 4.7):

1. Use the **Manage Materials** command on the **Advanced Simulation** toolbar.
2. In the *New Material* group, select *Isotropic* from the list and use the **Create Material** command.
3. In the dialog box, type a unique *Name* for the new material. Select the *Mechanical* tab, and in the *Elastic Constants* group, enter *Young's Modulus* and *Poisson's Ratio* values.
4. In the *Stress-Strain Related Properties*, specify the following parameters (Figure 4.7, A):
 - As the *Type of Nonlinearity*, select *PLASTIC*.
 - Select *Yield Function Criterion*, typically *von Mises*.
 - Specify the *Hardening Rule*, for example, *Isotropic*.
 - Enter *LIMIT1* for the *Initial Yield Point* parameter.
- 5a. If the bilinear method is used to define the strain diagram, in the *Stress-Strain* list, select *Expression* and enter the *H* value that specifies the

slope of the second linear segment.

5b. If piecewise linear method is used, in the Stress-Strain list, select (Figure 4.7, B):

- *Field*.
- A *Specify Field* parameter appears. Select *Table Constructor*.
- In the **Table Field** dialog box, select *Independent* tab and specify *Domain* type: *1D General* for simple strain relation, or *Temperature-Strain* for temperature dependent plastic properties.
- In the *Data Points* group, enter the tabular data of the deformation diagram. Follow the previously described rules and make sure you use correct units of measurement.
- Click **OK**.

6. Click **OK**. A new material appears in the list.

7. Click **Close**.

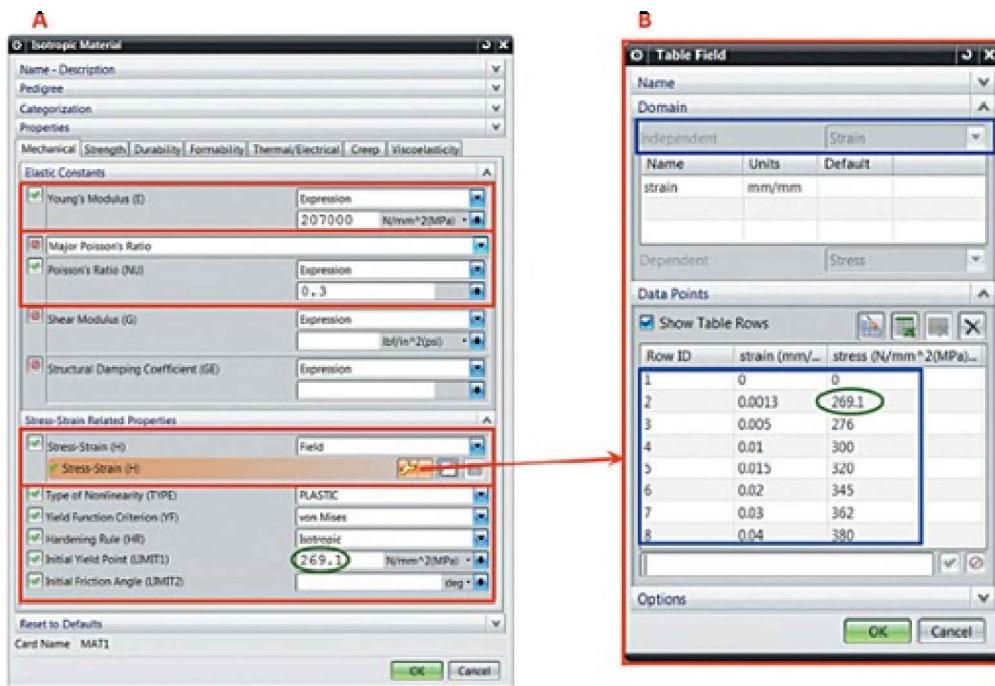


Figure 4.7. Specifying the stress-strain curve in tabular form

Depending on the nonlinearity type, the stress-strain curve is specified in engineering values or true values. With small strains (under 2%), the difference between these values is insignificant:

- In case of small displacements, small strains specify engineering

values.

- In case of large displacements, small strains, it is recommended to use true values.
- In case of large strains, true values must be used.

Engineering values can be converted into true values of the stress-strain curve with the CVSSVAL=1 parameter (Figure 4.8).

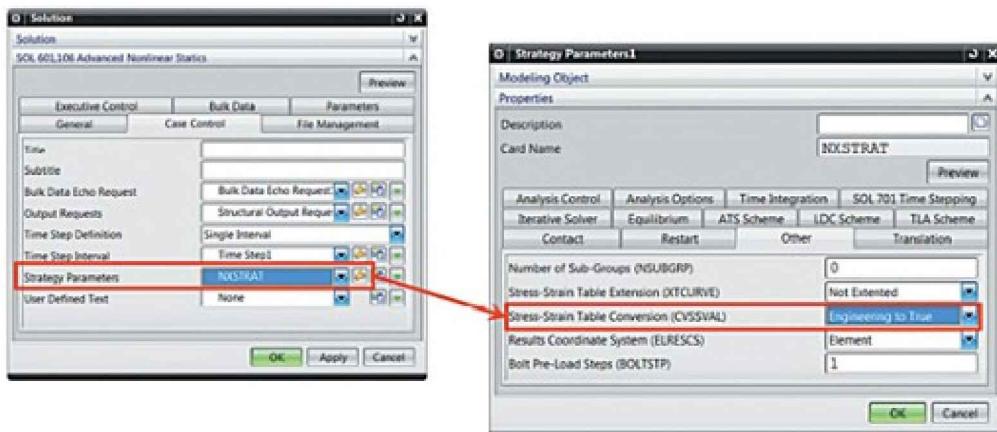


Figure 4.8. CVSSVAL parameter

To view information on plastic deformation in the structure for ADVNL 601, 701 solutions simply use NX Advanced Simulation post processor and display the distribution of plastic strain in the graphics area after solving the nonlinear problem. For the NLSTATIC 106 solution, plastic strain data are written into NX Nastran *.f06 file under the *Nonlinear Stresses* header. This file can be viewed in any text editor.

4.4. Contact interaction

Mechanical loading of deformable solid bodies most often includes contact interaction of two or more objects. Therefore, solid body loading analysis generally requires analysing contact interaction conditions. The loading conditions are often most severe in the contact zone, and structural failure processes often start at these surfaces. Contact interaction is a type of nonlinearity because it is accompanied by a change in the system's stiffness and a change of boundary constraints as the deformed state of the system changes (Figure 4.9).

In NX Advanced Simulation, you can model complex contact interactions

using SOL 601 and 701 solutions, which provide the following features:

- Contact of shell and solid-body faces of elements.
- Contact of edges for axisymmetric problems.
- Unilateral and bilateral contact.
- More than ten friction models.
- Accounting for frictional heating.
- Linked contact.
- Several solution algorithms for contact problems.
- Flexible contact and contact of rigid surfaces.
- Surface contact with offset.
- Nonlinear elements such as gaps.
- Output of results as contact pressures and forces.

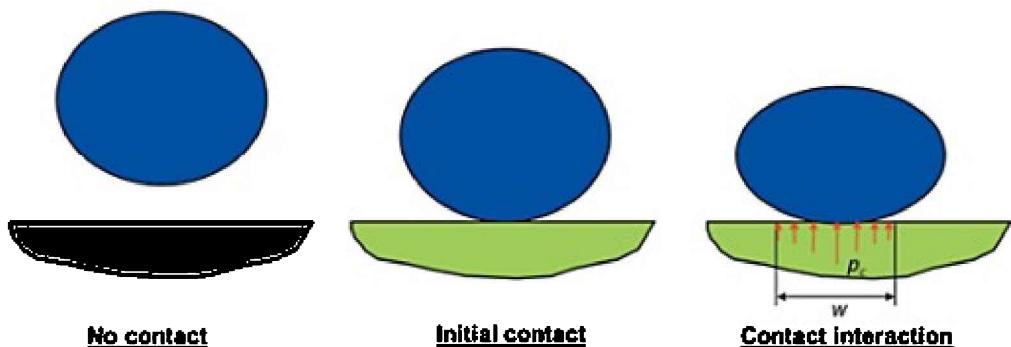


Figure 4.9. Contact interaction

4.4.1. Specifying contact interaction

Contact interaction (referred to as contact) is modelled using *contact sets*, *regions*, contact segments, and contact pairs (Figure 4.10). When you specify contact, you need to control the choice of the parameter set: *BCRPARA* for the contact region, *BCTP PARA* for the contact set. Parameters of strategy parameters *NXSTRAT* are additionally used. Contact constraints can be created for the simulated model using the **Surface-to-Surface Contact** command that is described in detail in Chapter 4 of Part 1.

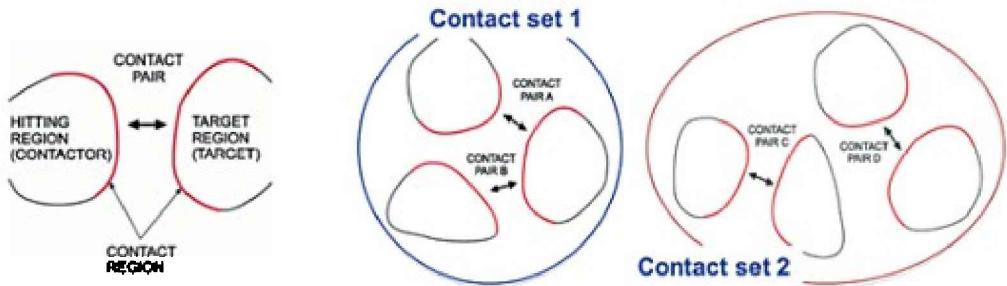


Figure 4.10. Contact sets, pairs, and regions

Modeling contact in NX Advanced Simulation and NX Nastran involves the following steps:

1. Create contact regions that can consist of a set of 3D element faces and 2D elements. These are the regions where contact is expected to occur. For algorithm execution, the regions are split into so-called contact segments. If necessary, you can specify *BCRPARA* parameters for the contact regions (Figure 4.11, A), for example, you can set a contact offset and specify the contacting side of a shell element.
2. Create contact pairs out of regions, then create contact sets that consist of contact pairs (Figure 4.10). A contact pair consists of two contact surfaces that may come in contact. One of the surfaces is called the contact, or source surface, and the other is called the target surface. In the solution of the problem, the nodes of the source surface and the segments of the target surface come in contact. The nodes of the source surface cannot penetrate the segments of the target surface, while the nodes of the target surface can. *BCTPARA* parameters of contact sets allow friction to be accounted for, and the selection of contact type, contact algorithm and much more (Figure 4.11, B). If the model contains several contact sets, each set can have its own *BCTPARA* parameters.
3. Specify the values of contact forces and pressure in the *Structural Output Requests* solution parameters to save these in the results file.
4. If necessary, edit the default contact parameters of the solution in the relevant tab of *NXSTRAT* (*Strategy Parameters*) parameter set.

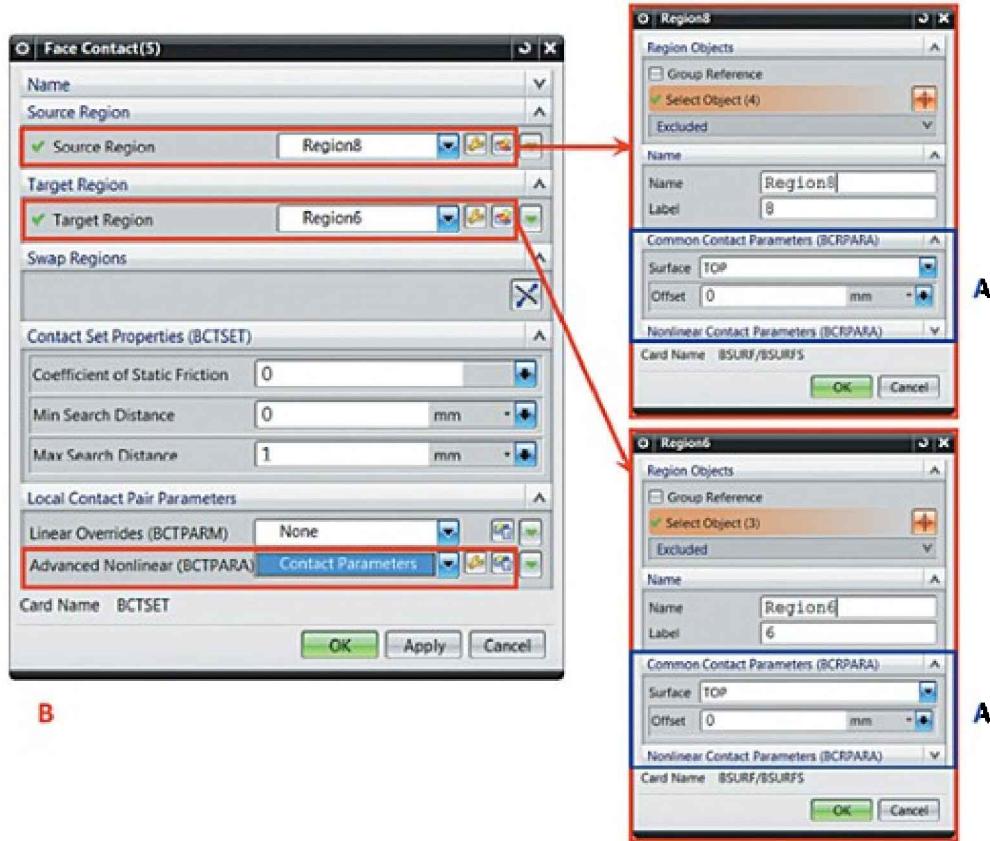


Figure 4.11. Contact interaction parameters

If there is contact interaction of a surface with itself (so-called self-contact), the same surface is selected as the contact and the target surface.

Sometimes, the target surface is made absolutely rigid (*BCTPARA* parameters) to reduce computation intensity in cases when one of the contacting parts is a rigid body. All nodes of the rigid surface must either have identical applied displacements or be rigidly bound to the so-called master node, which is defined in *BCTPARA* parameters. The displacement applied to the master node is uniformly transmitted to the nodes of the rigid surface.

If necessary, you can create symmetric contact pairs, that is, surface A can be the contact surface in pair 1 and the target surface in pair 2, while surface B can be the target surface in pair 1 and the contact surface in pair 2.

You should observe the following guidelines when modelling contact

interaction:

- Use four-node shell elements instead of eight-node elements.
- If you use 20-node hexahedral elements, convert these to 27-node elements using the *ELCV=1* solution parameter in the *NXSTRAT* parameter set (*Strategy Parameters*).
- Pairs of surfaces must be selected so that the nodes of the contact surface do not penetrate elements of the target surface.
- Select the surface with the coarser FE mesh as the target surface.
- If an edge contacts a surface, select the surface that contains the edge as the contact.
- If one of the surfaces has dependent degrees of freedom and displacement constraints, select it as the target surface.
- If one surface is significantly stiffer than the other, it should be the target surface.

4.4.2. Special aspects of contact problem solutions

Ideal contact constraints are of the following form (Figure 4.12, A):

$$g \geq 0; t_n \geq 0; g t_n = 0,$$

where t_n is the normal contact force, and

g is the gap between the contact and the target segment.

If contact occurs, $t_n > 0$ and $g = 0$. If there is no contact, $g > 0$ and $t_n = 0$.

Coulomb friction conditions are of the following form (Figure 4.12, B):

- In slipping, the ratio of tangential force t_t to normal force t_n is equal to the friction coefficient μ , while the direction of the tangential force is opposite the slip direction.
- In sticking, the tangential velocity is zero, and $t_t < \mu t_n$.

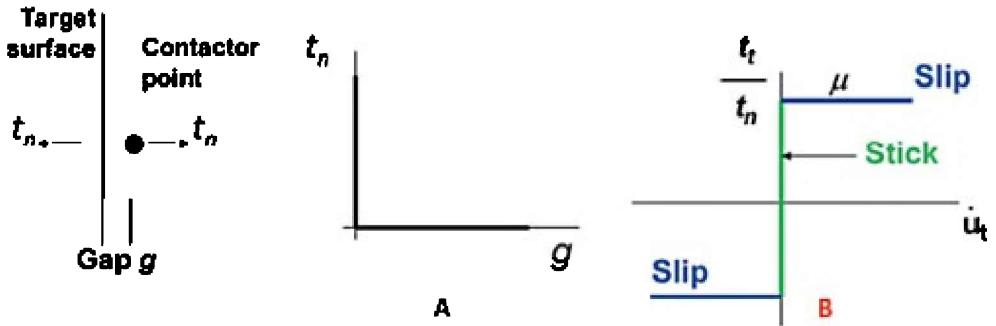


Figure 4.12. Ideal contact constraints

Algorithms for satisfying contact constraints

Strength analysis problems with contact interaction are characterized by contact boundary constraints that must be satisfied at the contacting surfaces. There are the following methods of satisfying contact boundary constraints in ADVNL 601,106/129 solutions [23]:

1. *Constraint Function Method* is the most general method. It works well in most cases and is used by default. The main concept of this method is the introduction of a constraint function to replace ideal contact constraints with non-ideal constraints (Figure 4.13). To set it, specify *TYPE* = 1 in the *BCTPARA* contact parameter set.
2. *Segment Method* is sometimes used in problems that involve contact with friction. This method is used with ideal contact constraints. To set it, specify *TYPE* = 2 in the *BCTPARA* parameter set.
3. *Rigid Target Method* is a specialized method for stamping problems. To set it, specify *TYPE* = 3 in the *BCTPARA* parameter set.

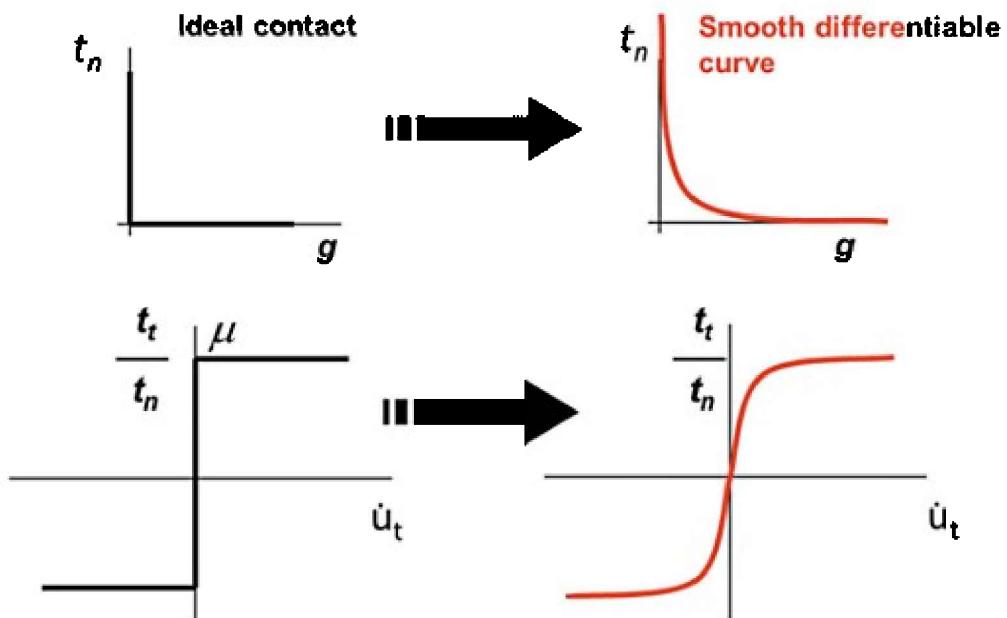


Figure 4.13. Replacing ideal contact constraints with non-ideal constraints

There are three contact methods for the ADVNL 701 solution:

1. *Kinematic Constraint Method* is the most precise. It is used in most cases. This method guarantees the absence of penetration or oscillation. To set it, specify *XTYPE* = 1 in the *BCTPARA* parameter set.
2. *Penalty Method* is the simplest and fastest, it works in most cases. To set it, specify *XTYPE* = 2.
3. *Explicit Rigid Target Method* is a specialized method for stamping problems. To set it, specify *XTYPE* = 3.

Note that you can specify different contact algorithms for different contact pairs of the same solution.

If you use the restart ability, that is, if you restart the calculation with results of a converged step taken into account, you can change solution types and contact algorithms. However, when you switch from ADVNL 601 solution to ADVNL 701 solution, you can only replace the *Rigid target method* with the analogous *Explicit Rigid Target Method* and vice versa.

Contact interaction with friction uses the Coulomb friction law. The friction coefficient μ can be a constant value or a variable that is calculated during the solution process using a certain law (for ADVNL 601 only). You can use

the following variables: relative slip velocity, displacement, contact forces, time, and coordinates.

A constant friction coefficient is used by default. It can be set in the *BCTSET* parameters. There are several friction models with different laws [23] that can be set with the *FRICMOD* parameter in the *BCTPARA* set of contact parameters (Figure 4.11, B).

4.5. Solving nonlinear problems

Nonlinear problems can be solved in NX Nastran Advanced Nonlinear using stable algorithms. For most complex and non-trivial nonlinear problems, it is quite possible to achieve solution convergence.

The following is a comparison of linear and nonlinear static problem solutions.

SESTATIC 101 linear static solution:

- Calculates the element stiffness matrix $[K]$ once at the beginning of the solution.
- Uses Hooke's law to calculate nodal displacements.
- Does not account for large displacements and rotations.
- Does not update load directions when the deformed state changes.
- Nonlinear NLSTATIC 106 and ADVNL 601, 106 solutions:
- Iteratively track the nonlinear force-displacement relation.
- Periodically update the stiffness matrix.
- Account for large displacements and rotations.
- Track load directions when the deformed state changes.
- ADVNL 601, 106 executes the contact algorithm at each iteration.

You can solve nonlinear static and implicit dynamic problems using the following methods [23]:

- Newton-Raphson iteration method (equilibrium iteration method), which can be applied to most problems with and without the line search option.
- Arc length method, also known as the displacement control method (LDC), which is used for problems with post-buckling

behaviour.

In implicit dynamic analysis, time integration is carried out using the method specified in the *TINTEG* parameter of the *NXSTRAT* set:

- Newmark method, used by default (*TINTEG* = 0). This method is based on the trapezoid rule.
- Composite method (*TINTEG* = 1), which comprises two steps: the trapezoid rule is used for the first step, while the second step is based on the reverse Euler method.

4.5.1. NLSTATIC 106 and 129 solution parameters

If you use NLSTATIC 106 and 129 nonlinear solutions, the following parameters are key:

- Select the *Large Displacements* check box to account for large displacement effects (Figure 4.14, A) for static and dynamic solutions.
- In the *Number of Increments* box, type the number of load increments that is used for an incremental application of the load to the structure in a static solution. If you encounter unsatisfactory convergence, increase this value (Figure 4.14, B).
- In the *Intermediate Output* list, select whether results should be written for all converged steps or only for the last one (Figure 4.14, B).
- In the *Number of Time Steps* box, type the number of time integration steps for dynamic analysis (Figure 4.14, C).
- In the *Time Increment* box, enter the time integration step (Figure 4.14, C).
- In the *Interval for Output*, set up results recording at every *n*-th step (Figure 4.14, C).

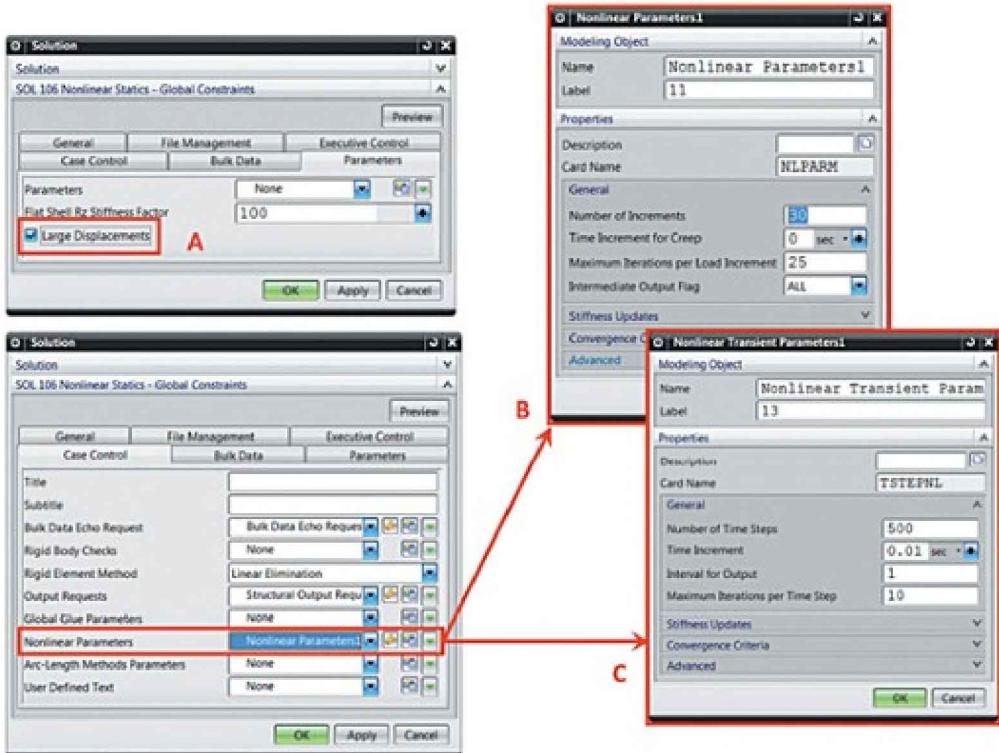


Figure 4.14. Key parameters of NLSTATIC 106 and NLTRAN 129 solutions

4.5.2. Parameters of ADVNL 601 and 701 solutions

When using ADVNL 601 and 701 nonlinear solutions, the following parameters are key:

- *Large Displacements* allow geometrical nonlinearity to be accounted for. This parameter can be found in the *Parameters* tab of the **Solution** dialog box (Figure 4.2, B).
- In the *Time Step Definition* list, specify one (by default) or several time intervals to be created (Figure 4.15, A).
- Under *Time Step Interval*, specify the number of time increments, the magnitude of the increment, and the results recording frequency (Figure 4.15, A).
- Under *Strategy Parameters*, you can specify a number of parameters (**Strategy Parameters**) that are paramount to a successful nonlinear analysis. The parameters belong to several types depending on their role in nonlinear simulation. There is a separate tab for each type (Figure 4.15, B):

- In the *Analysis Control* tab, you can select the solver and time increment scheme.
- In the *Analysis Options* tab, you can specify parameters that are responsible for stabilizing the stiffness matrix, u/p formulations of elements, maximum displacement limits, and follower load tracking.
- In the *Time Integration* tab, you can select the time integration method and its parameters.
- In the *Translation* tab, you can raise the order of the elements, and select the behaviour of RBAR/RBE2 elements.
- In the *Other* tab, you can specify parameters that are responsible for interpreting stress-strain curves of materials, bolt pretension, and so on.
- In the *Restart* tab, you can specify parameters that are relevant to restarts.
- In the *Contact* tab, you can specify a set of parameters to improve convergence of contact problems.
- In the *ATS Scheme*, *LDC Scheme*, and *TLA Scheme* tabs, you can specify parameters that correspond to the various time step selection schemes. You can select the time step variation scheme in the *Analysis Control tab*.
- In the *Equilibrium* tab, you can select Newton-Raphson method parameters.
- In the *Iterative Solver* tab, you can set parameters that are used in selecting the system of equations for the *3D Iterative Solver*.
- In the *SOL 701 Time Stepping* tab, you can specify a set of time step selection parameters for explicit dynamic analysis.

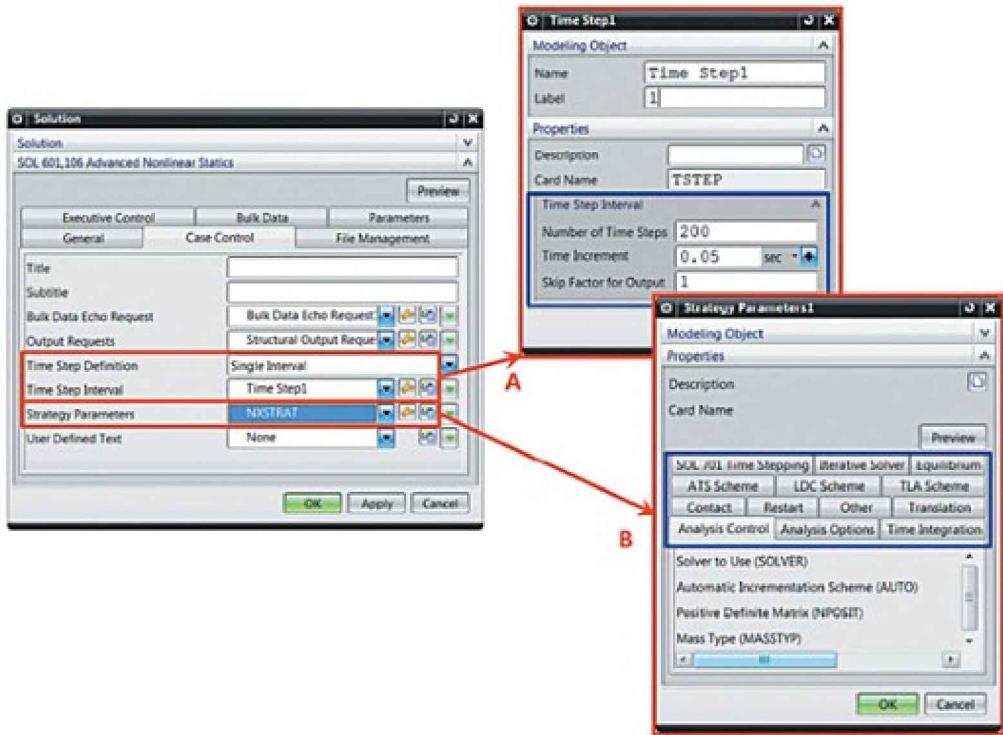


Figure 4.15. Key parameters of ADVNL 601 and 701 solutions

4.5.3. Additional features

Restart is an important addition to NX Nastran Advanced Nonlinear, because it allows the solution of a problem to be stopped and restarted with a different analysis type or different loads and boundary conditions. During a restart, you can switch from static analysis to dynamic analysis and vice versa. You can also modify analysis and solution options, such as time integration step, contact interaction parameters, material properties, and so on.

You can remove and add any supported elements at any given time during the solution of the nonlinear problem. You can also delete elements whenever a limit point of the elastoplastic material stress-strain curve is reached. If an element is removed from the model, its mass and stiffness are removed gradually to improve convergence of the solution.

NX Nastran Advanced Nonlinear provides advanced coupled analysis of thermo elastic problems either with mutual influence of thermal and structural solutions, or with one-way influence of the thermal state on the

structural analysis results. In addition to temperature strain, coupled thermo elastic analysis also accounts for the following effects:

- Internal heat generation due to plastic deformation of the material.
- Heat transfer between contacting bodies.
- Heat generation in the analysis of contact interaction with friction.

4.6. Example. Analysis of thin-sheet stamping

In this example you create a simulation model and simulate the nonlinear stamping process for a thin aluminium sheet (Figure 4.16). You will use the ready-made *Shtamp_fem.fem* FE model file, the *Shtamp_sim.sim* simulation model file, and *Stress-strain.csv* file, which contains tabular data of the stress-strain diagram for the elastoplastic material of the sheet. The objective of this nonlinear simulation is finding the load that must be applied to shape the sheet blank as necessary. The solution of the problem is complicated by the following nonlinearities:

- Contact interaction with friction.
- Elastoplastic material properties.
- Large displacements and large strains.

The example includes the following steps:

1. Open the simulation model and the FE model, specify the stress-strain curve of the elastoplastic material.
2. Create contact interaction constraints.
3. Specify boundary constraints.
4. Create the nonlinear solution and set up parameters.
5. Review the results.

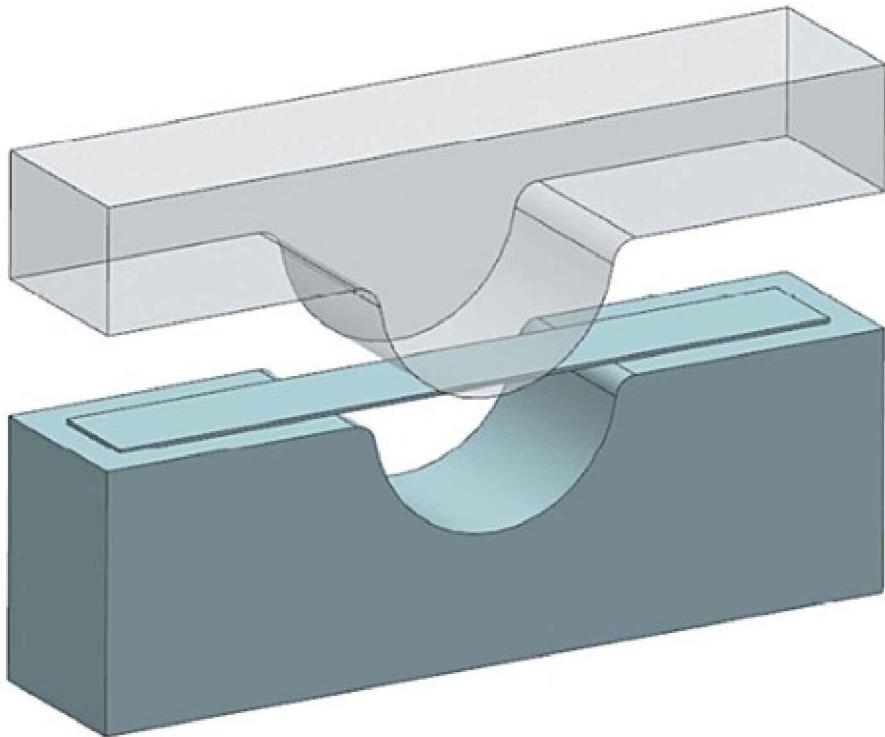


Figure 4.16. Geometry model of the die

The proposed FE model is a collection of shell elements and rigid elements that describe the symmetrical part of the structure (Figure 4.17, A). It is assumed that massive parts of the structure, the male die and the female die, are rigid. Contact constraints are set up with this assumption in mind. It is assumed that all nodes of the male die have the same displacement, therefore, enforced displacement is applied to a single node that is linked to all nodes of the movable rigid part through RBE2 elements.

Symmetry conditions and vertical offset conditions of the sheet are specified as boundary constraints (Figure 4.17, B).

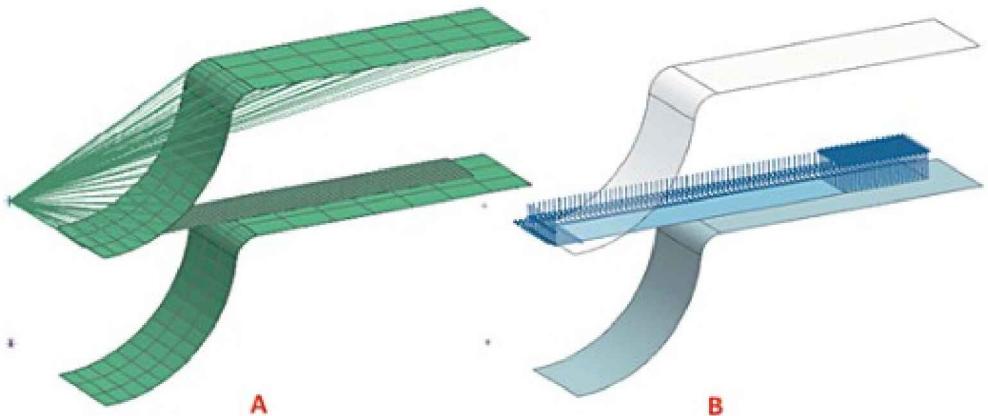


Figure 4.17. A. FE model of the die. B. Simulation model of the die

4.6.1. Specifying the stress-strain curve of the elastoplastic material

Launch NX Advanced Simulation and open the *Shtamp_sim.sim* file. Go to the *Shtamp_fem.fem* model.

Note that the model tree includes two 2D mesh collectors:

- *Matrix_Puanson* with a set of shell finite elements with dummy physical properties.
- *Zagotovka* with a shell mesh for the thin sheet.

Edit the properties of the sheet material by adding plastic properties. Do the following:

- Double-click *Zagotovka* to open the **Mesh Collector** dialog box.
- Use the **Edit** command for the *Shell Property* parameter.
- In the new dialog box, use the **Select Material** command.
- Select *Local Materials* in the *Material List*, then right-click the *Mat2* material and choose **Edit** in the shortcut menu.
- In the new dialog box, make sure that in the *Stress-Strain Related Properties* parameter group, in the *Type of Nonlinearity* list, you select *PLASTIC*.
- In the *Stress-Strain (H)* list, select *Field*.
- Under *Specify Field*, use the **Table Constructor** command.
- In the new table creation dialog box, in the *Domain* list, select *1D General*.

- In the *Data Points* group, use the *Import from file* command and select the existing *Stress-strain.csv* file, which contains stress-strain deformation curve data for the material in tabular form. Click **OK**.
- Enter a value of *181.7 MPa* for the *Initial Yield Point* parameter.
- Click **OK** in all dialog boxes.

4.6.2. Creating contact interaction constraints

Go to the simulation model of the *Shtamp_sim.sim* file and use the **Surface-to-Surface Contact** on the Advanced Simulation toolbar.

Create a contact pair as shown in Figure 4.18. When you specify the *Target Surface* in the *Region* dialog box, as the nonlinear contact parameters (*BCRPARA*) select *Type – RIGID*. Use the **Select Master Grid Point** command and specify the node at the origin point (Figure 4.18).

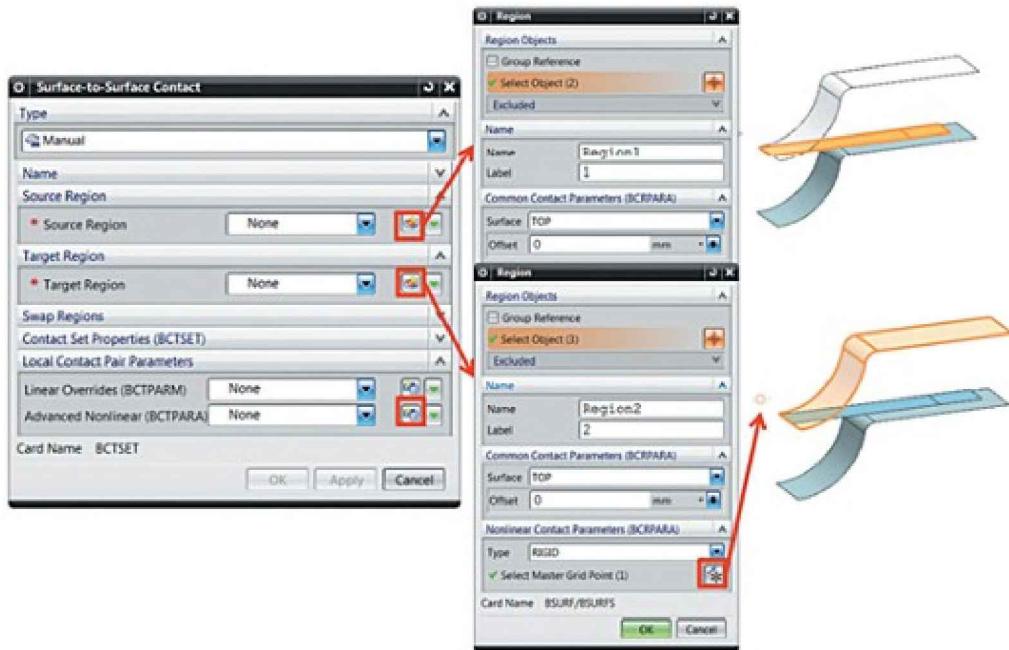


Figure 4.18. Specifying surface-to-surface contact conditions for the male die and the blank

Do not close the **Surface-to-Surface Contact** dialog box. Under **Advanced Nonlinear (BCTPARA)**, use the **Create Modeling Object** command. In the new dialog box, specify the following contact interaction parameters:

- In the *Type of Offset (OFFSET)* list, select *Half the Shell Thickness*.
- In the *Friction Model Type (FRICMOD)* box, enter 4 to use the corresponding friction model.
- For the *Friction parameter A1 (FPARA1), ... A2 (FPARA2), and ... A3 (FPARA3)*, enter 0.2, 0.1, and 2e-4 respectively.

Click **OK** in all dialog boxes.

In a similar way, create the second contact pair as shown in Figure 4.19. Note that when you specify a *Source Surface*, in the *Surface* list, you need to select *Bottom*. When you specify the *Target Surface* in the dialog box of the *Type* parameter, select *RIGID*. Use the **Select Master Grid Point** command and specify the existing node with coordinates (0,0,-25). As the *Advanced Nonlinear (BCTPARA)* parameter set, select the created set *Contact Parameters – Advanced Nonlinear Pair 1*.

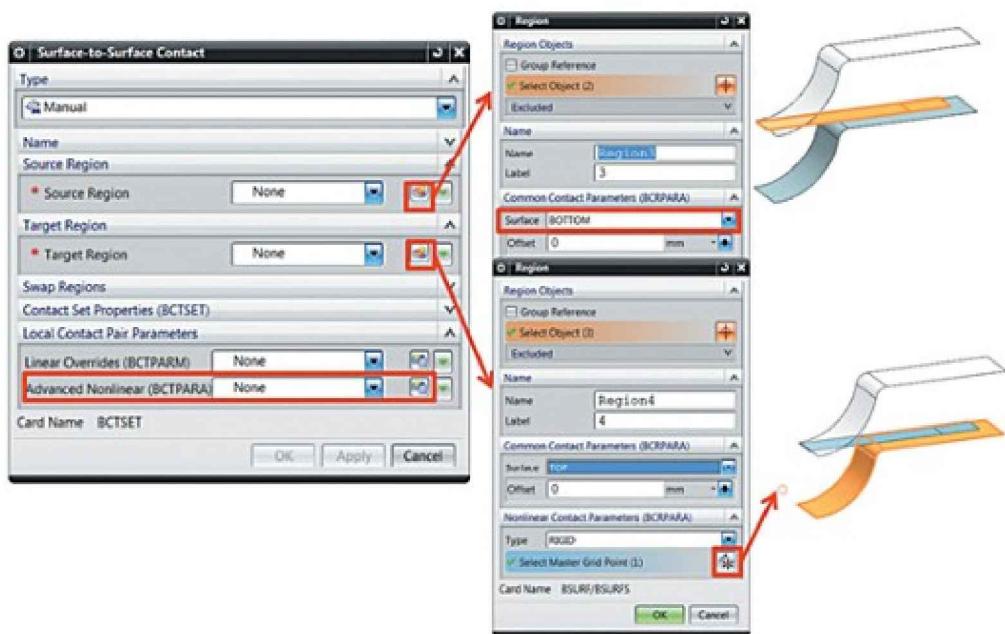


Figure 4.19. Specifying surface-to-surface contact conditions for the blank and the female die

4.6.3. Specifying boundary conditions

To constrain all degrees of freedom of the male die, which is absolutely rigid with a master node assigned, select the **Fixed Constraint** command from

the **Constraint Type** list on the **Advanced Simulation** toolbar and select the node with coordinates (0,0,-25) (Figure 4.20). Click **OK**.

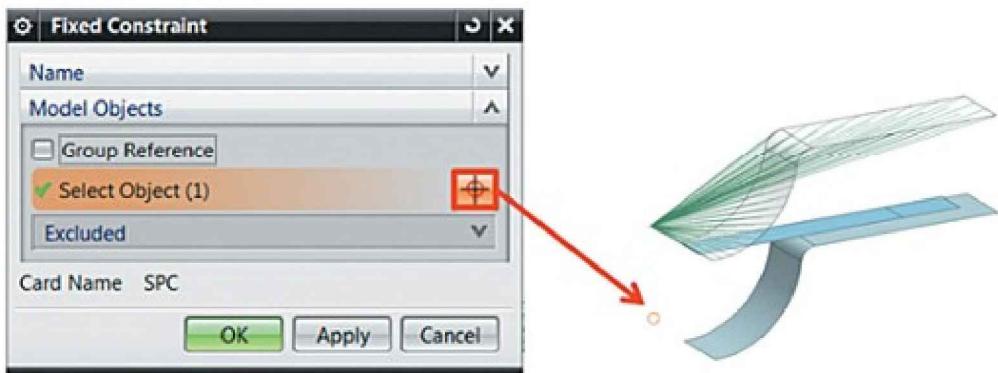


Figure 4.20. Constraining displacement of the female die

To specify the gradual displacement of the rigid male die, use the **Enforced Displacement Constraint** command in the **Constraint Type** list on the **Advanced simulation toolbar**. Select the master node of the male die with coordinates (0,0,0) (Figure 4.21). For *DOF1*, *DOF2*, *DOF4*, *DOF5*, *DOF6* parameters, enter zero values. For *DOF3*, select *Field* and create a data table: as the independent variable of the *Domain* parameter, select *Time*. In the *Data Points* group, specify values as shown in Figure 4.21. Click **OK** in all dialog boxes.

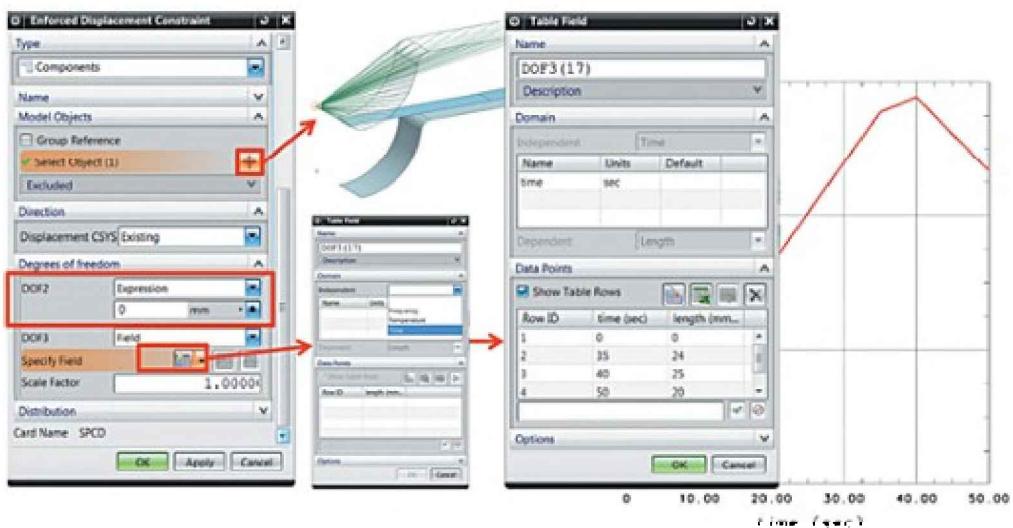


Figure 4.21. Setting up enforced displacement for the male die

4.6.4. Creating the nonlinear solution

In the shortcut menu of the *Shtamp_sim.sim* node, choose the **New Solution** command.

In the dialog box, specify the parameters of the nonlinear solution:

- As the *Solution Type*, select **ADVNL 601,106**.
- Select the *Case Control* tab.
- Under *Output Request*, use the **Edit** command.
- In the new dialog box, select the *Contact Result* tab and select the *Enable BCRESULTS Request* check box.
- Select the *Nonlinear Stress* tab and select the *Enable NLSTRESS Request* check box.
- Click **OK**.
- Under *Time Step Interval*, use the **Create Modeling Object** command.
- In the new dialog box, as the *Number of Time Steps*, type **50**.
- Click **OK**.
- Under *Strategy Parameters*, use the **Create Modeling Object** command.
- In the new dialog box, select the *Analysis control* tab, and in the *Automatic Incrementation Scheme (AUTO)* list, select **ATS**.
- Select the *ATS Scheme tab*, and in the *Smallest Time Step Size Number (ATSSUBD)* box, enter **1000**.
- Click **OK**.
- Select the *Parameters* tab, and select the *Large Strains* check box.
- Click **OK**.

In the simulation model tree, drag all created constraints and simulation objects to the new solution. To resolve degree of freedom conflicts, right-click the *Solution 1* node and choose the **Resolve Constraints...** command. In the conflict dialog box, right-click *Keep Overlapping*, that is, keep all constraints. Click **OK**.

To run the simulation, right-click the *Solution 1* node and choose the **Solve...** command. Click **OK**.

Note that in the new **Solution Monitor** dialog box (Figure 4.22) the **Nonlinear History** tab shows a graph that displays the current iteration

number and time in real time.

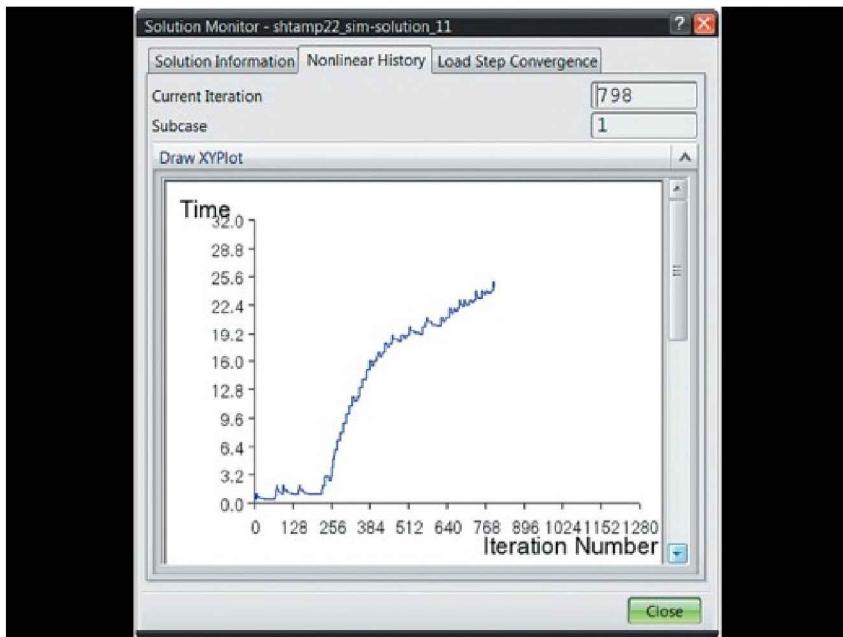


Figure 4.22. The *Solution Monitor* dialog box

4.6.5. Viewing the nonlinear analysis results

Load *Solution 1* results in the **Post Processing Navigator** tab. Display the nodal distribution of total displacement for any of the *Non-Linear Steps*.

To set up the display of full-scale deformed state, use the **Edit Post View** command on the **Post Processing toolbar**. In the **Post View** dialog box, use the **Results** command next to the *Deformation option*. In the new dialog box, in the *Scale* list, select *1.0 Absolute*. Click **OK** in all dialog boxes.

To create an animation, use the **Animation** command of the **Post Processing** toolbar. In the new dialog box, in the *Animate* list, select *Iterations*. Click **OK**.

Display the von Mises residual plastic strain distribution that persists after the load is lifted (Figure 4.23). To do so, go to the results of the last nonlinear step, double-click the *Nonlinear Strain – Nodal/Elemental*.

Nonlinear Strain - Element-Nodal, Averaged, Von-Mises

Shell Section : Top

Min . 0.000, Max : 0.103, Units = mm/mm

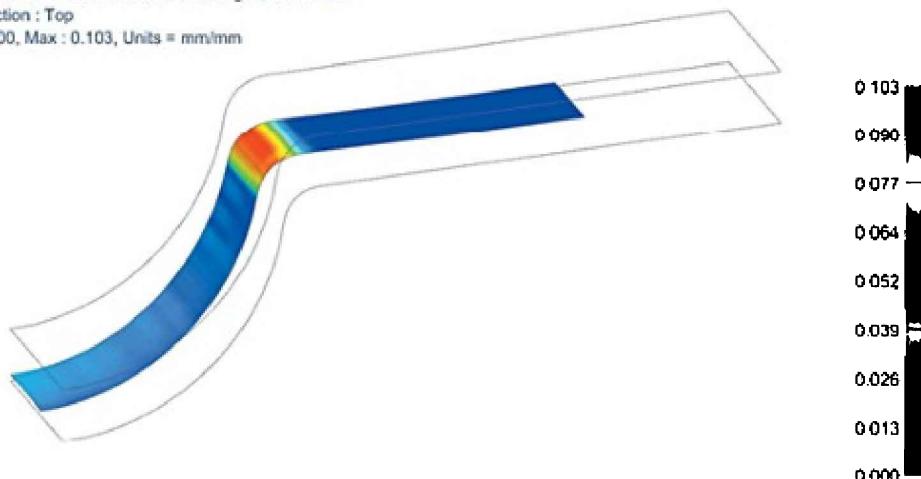


Figure 4.23. Von Mises strain distribution in the sheet

Display reaction forces by selecting the Z component in the *Reaction Force – Nodal* node of the results tree for the fortieth nonlinear iteration. To visualize the reaction with arrows (Figure 4.24), use the **Edit Post View** command. In the **Post View** dialog box, in the *Color Display* list, select *Arrow*.

Contact Force - Nodal, Magnitude

Min : -0.943, Max : 0.005, Units = N

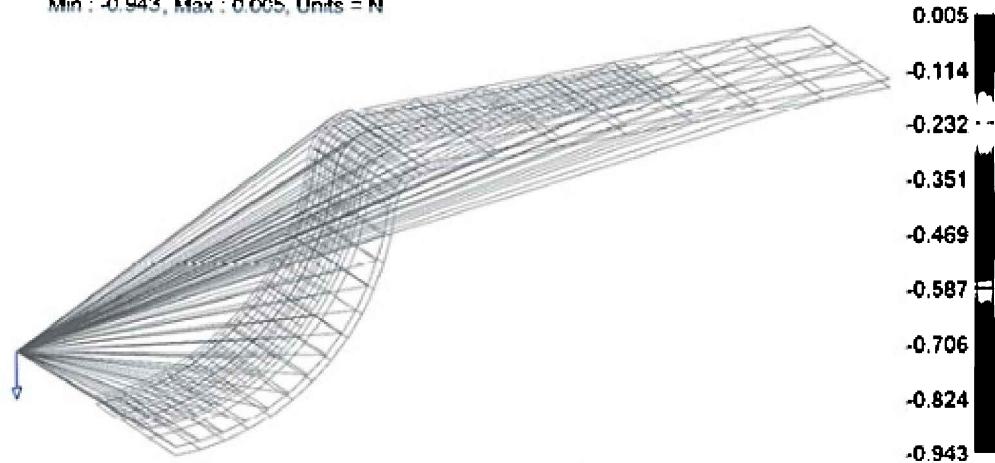


Figure 4.24. Z component of the reaction force at the 40th iteration

Right-click the *Post View n* node of the results tree and choose the **New Graph...** command. In the new dialog box, in the *Graph Type* list, select *Across Iterations*, for the *Node ID* parameter, select the master node of the male die (Figure 4.25). Click **OK**. The graphics area plots the change of the

relevant component of the reaction force that arises when the male die pushes the thin sheet into the female die.

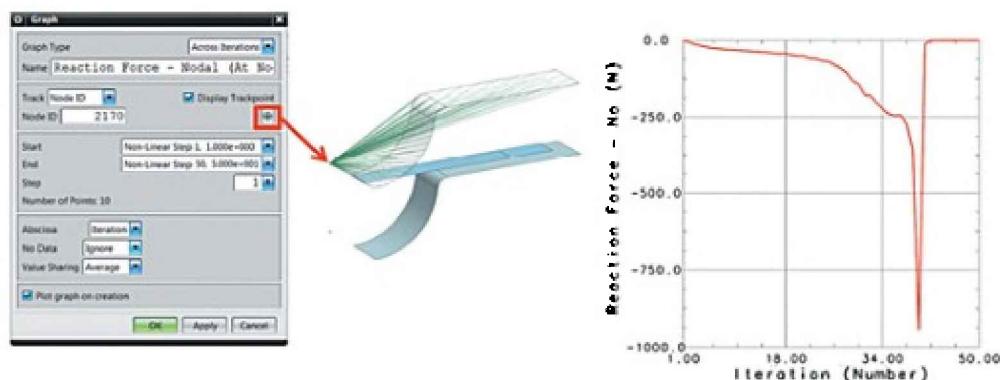


Figure 4.25. Variation graph of the Z component of the reaction force at the master node of the male die

Chapter 5. Analysis of heat and mass transfer

Designing a state-of-the-art product is complex, multifaceted work, which must take into account many different processes and phenomena. One such process that occurs in most devices is generation of thermal energy. Production of heat is often the end purpose of the process, or sometimes a parasitic phenomenon. In both cases it needs to be analyzed.

NX Thermal/NX Advanced Thermal modules provide a broad spectrum of heat transfer analysis features for R&D engineers. The algorithms implemented in these modules can account for heat transfer due to thermal conductivity, convection, and radiant heat exchange. These heat transfer mechanisms can be used separately or altogether. The solver works with one, two, three, and zero-dimensional meshes, allowing frugal use of available computing resources. The approaches used by NX CAE modules can also describe heat exchange with the environment without directly modelling it. This is achieved by defining specific boundary conditions.

5.1. Capabilities of the heat transfer analysis module

Well-developed mathematics, diversity of physical models, and the combination of engineering and numerical approaches are the foundations of NX Thermal and allow the effective formulation and solving of complex technical problems. The features of the application that determine its applicability based on whether particular physical and mathematical models are implemented, are divided, from the licensing point of view, into basic (NX Thermal) and advanced (NX Advanced Thermal) features.

The basic features include:

- Solving heat transfer problems in steady-state and transient formulations
- Simultaneous processing of thermal conductivity and radiation mechanisms
- Modelling heat transfer between disjoint or multiple separate objects (also in the context of assemblies)
- Creating calculation schemes with control elements (thermostats and heat controllers).

- Accounting for radiant heat exchange using the Hemicube method (a video card can be used for computation)
- Creating multidisciplinary analysis types
- Full integration with the fluid dynamics solver
- Linear and nonlinear material properties
- Spatially varying boundary conditions
- Transient boundary conditions.

If necessary, you can significantly extend the feature set of the system by utilizing the Advanced Thermal module. In this case you gain the following features:

- Accounting for convective heat exchange (free convection and forced convection).
- Advanced thermo-optical properties of materials.
- Additional models and advanced settings for radiant heat exchange.
- Film cooling.
- Accounting for solar radiation.
- Accounting for various radiation sources.
- Radiation of nongray bodies.
- Accounting for motion of modelling objects: translation and/or rotation.
- Accounting for heating due to electric current (Joule heating), Peltier cooler elements.
- Additional heat controller models.
- One-dimensional flow in ducts.
- Heat transfer in composite materials based on laminar plates.
- Modeling of ablation and charring.
- Open architecture: support of custom subroutines.
- Parallel computing and some other features.

The rest of the chapter describes the ***Advanced Thermal*** module, which is a superset of the basic ***Thermal*** module. Some of the discussed features may thus not be available if only the ***Thermal*** module license is used.

5.2. Operating principle

The standard stages of solving a problem using the finite element method

are: preparing the model (pre-processing), running a solve, and processing of results (post processing). In general, these stages can be distinguished in heat transfer problems as well, however, each stage features its own specific settings and mandatory operations. The longest and hardest is typically the model preparation step. It can be subdivided as follows:

- Constructing the geometry of the simulation region (or idealizing the design model)
- Constructing the simulation mesh (building the FE model)
- Formulating the problem (specifying material properties, boundary conditions, choosing mathematical models, and other aspects)

To use **NX Advanced Thermal**, like any other application that is based on the finite-element method, you need a mesh model. The basic principles and methods of creating FE models are discussed in Chapter 4 of Part 1. Two special aspects of material properties are relevant for thermal problems: thermo-optical properties and multi-layer materials.

5.2.1. Specifying thermo-optical properties for the material

Correctly describing heat transfer with radiant exchange requires specification of thermo-optical properties of participating materials. Basic and advanced thermo-optical properties are available. You can set thermo-optical properties at the level of the *FEM* file using the **Modeling Objects** command on the **Advanced Simulation** toolbar. After clicking this command, in the **Modeling Objects Manager** dialog box, in the *Type* list, select *Thermo-Optical Properties/Thermo-Optical Properties - Advanced* and click *Create* to confirm the selection. The objects you create are added to the *Selection* list as shown in Figure 5.1, A. If you select basic thermo-optical properties, you can only specify the *Emissivity* of the surface (Figure 5.1, B). If you need to account for properties like *Absorptivity*, *Specular Reflectivity*, *Index of Refraction*, or *Transmissivity*, you need to set up advanced thermo-optical properties. In addition, you can set properties for infrared and visible spectra separately (Figure 5.1, C).

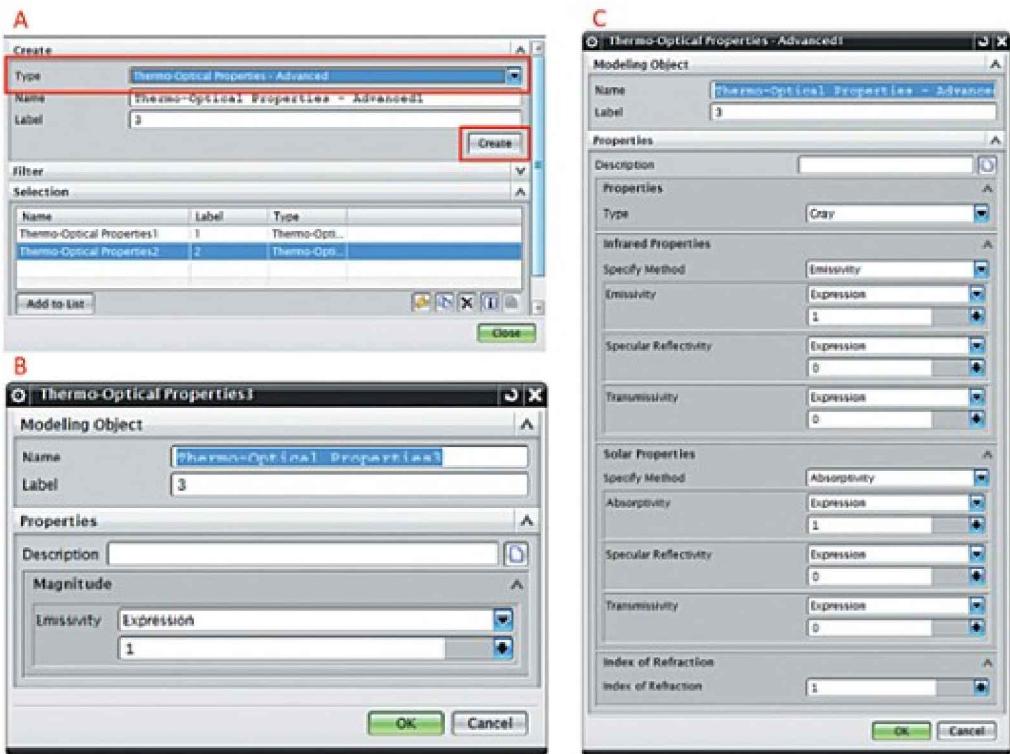


Figure 5.1 Creating a modeling object: specifying standard and advanced thermo-optical properties of the material

5.2.2. Creating multi-layer materials

One of the numerical modelling tools that allow significant simplification of problems without losing the accuracy of its formulation is the creation of two-dimensional *Layer* objects. You can select a *Layer* type modelling object in the same way you would select thermo-optical properties.

When you create a *Layer* object, you need to set up the following properties:

- *Material*: physical properties of the material that describes the layer.
- *Thickness*: the thickness of the material that describes the layer.
- *Thermo-Optical Properties*: properties that describe the material's radiant heat exchange parameters.
- *Coupling to Layer above*: parameters of thermal coupling between layers in multi-layer materials.

You can use the *Layer* object to describe solid bodies using surface mesh

models (in cases where such representation is appropriate). A single surface mesh model can contain many *Layer* objects in any combination. This allows you to create models of multi-layer (lamine) materials that reflect the variation of properties within a single layer, and layer-to-layer variation. You can create custom parameters of heat exchange between individual layers and specify thermo-optical properties separately for each layer.

Material properties are usually described at the simulation mesh creation stage in the *FEM* module.

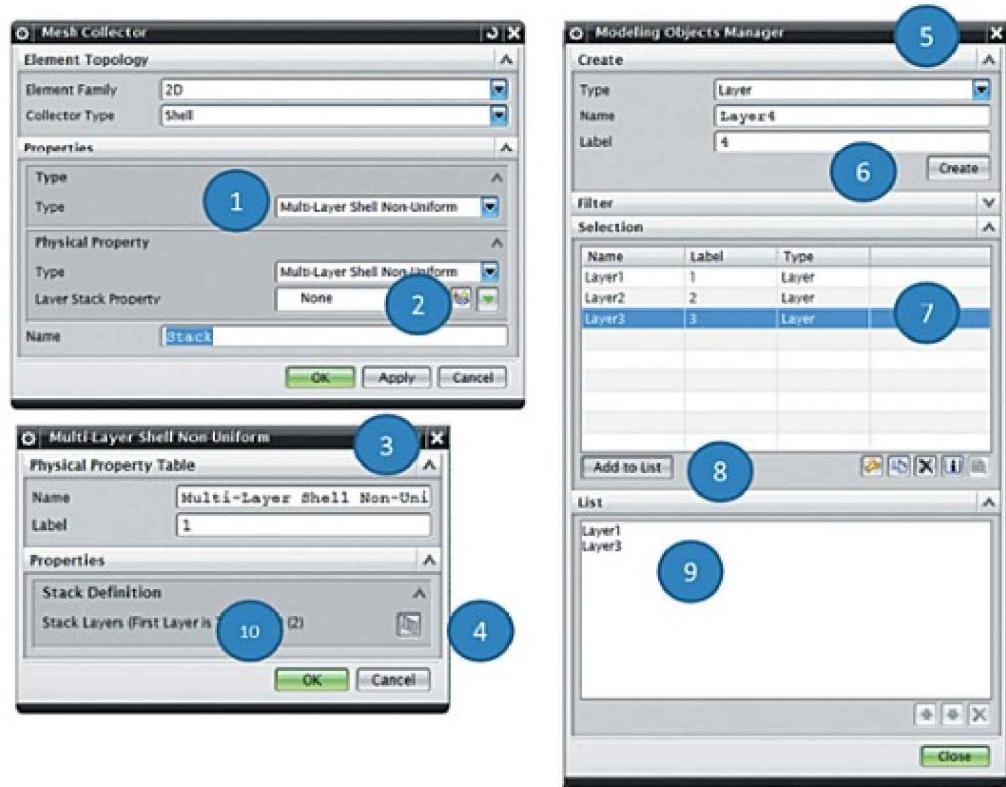


Figure 5.2. Multi-layer object creation procedure

Here is the procedure for creating a non-uniform multi-layer shell.

1. Select *Multi-Layer Shell Non-Uniform* in the *Type* list of the **Mesh Collector** dialog box (Figure 5.2, 1).
2. Click *Create Physical* to open the **Multi-Layer Shell Non-Uniform** dialog box (Figure 5.2, 2 and 3).
3. Click *Create Stack Layers* (Figure 5.2, 4) to open the **Modeling Objects Manager** dialog box (Figure 5.2, 5).

4. Click **Create** to open the layer creation dialog box (Figure 5.2, 6).
5. Specify the layer properties, including the parameters of thermal coupling with the adjacent layer and click **OK**.
6. Go to the **Modeling Objects Manager** dialog box and return to the created object in the **Selection** group (Figure 5.2, 7).
7. Repeat steps 4 through 6 to create a set of layers.
8. Click **Add to List** in the **Modeling Objects Manager** dialog box as shown in Figure 5.2, 8, then select relevant layers and create a sequence for them. This sequence is displayed in the **List** box (Figure 5.2, 9).
9. Click the **Close** button to return to the **Multi-Layer Shell Non-Uniform** dialog box. In the **Stack Definition** box a number in parentheses indicates the number of layers in this collector (Figure 5.2, 10). To modify the stack, click **Create Stack Layers** (Figure 5.2, 4). Click **OK** in the **Multi-Layer Shell Non-Uniform** to finish the multi-layer material creation and return to the **Mesh Collector** dialog box.

If you have an FE model and a set of defined material properties, you can now set up the boundary conditions for the problem. You can, of course, modify the material properties and the FE mesh as you continue to formulate the problem.

5.3. Tools for specifying boundary conditions

The following is a description of available tools, their capabilities and usage scenarios. Since all tools described are only available in the corresponding application, it makes sense to begin with the launching procedure and initial setup of the solver and solution.

5.3.1. Starting the application

Create a new solution for thermal analysis: in the **Simulation Navigator** tab, right-click the topmost node of the model tree and choose **New Solution** if **SIM** file level is currently selected, or **New Simulation** if **FEM** file is selected. In the dialog box (Figure 5.3), as the **Solution Type**, select **Thermal** or **Advanced Thermal**.

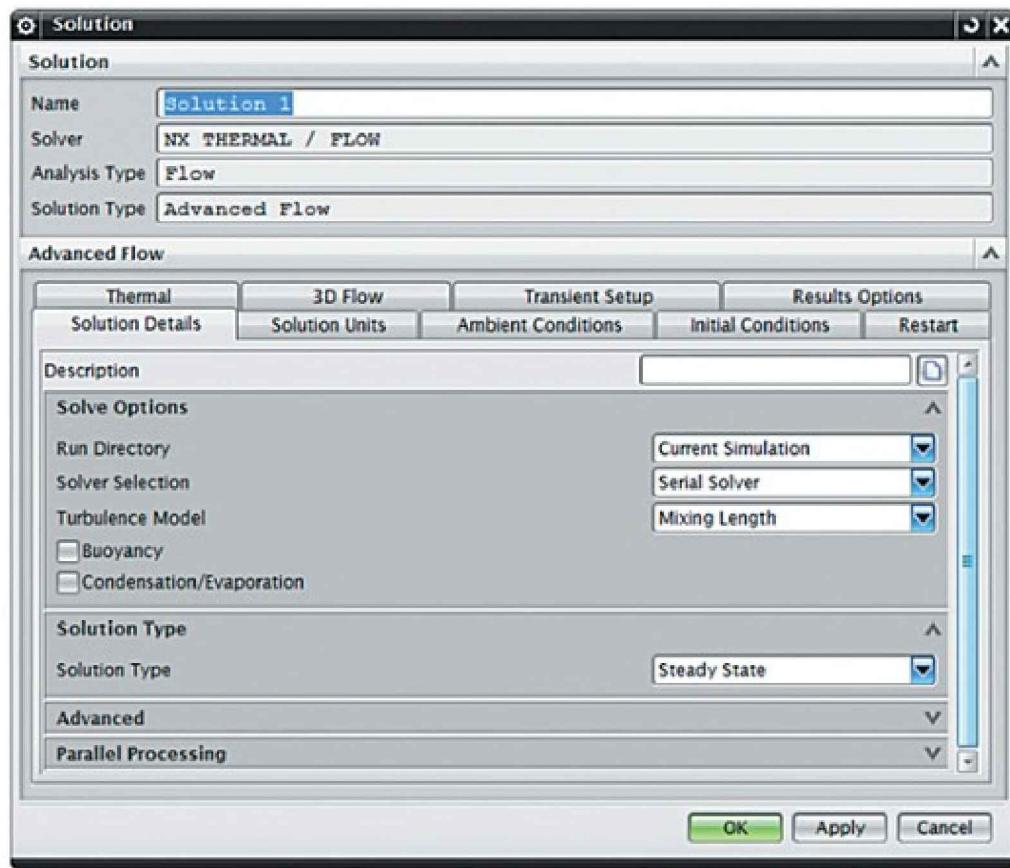


Figure 5.3. Creating a solution for thermal analysis

The **Solution** dialog box has a number of tabs that you can use to set up solver parameters.

You can use the *Solution Details* tab to set up the basic parameters of the problem (analysis type and solution method):

- In the *Run Directory* list, you can set up the location of files needed to run the simulation.
- In the *Solution Type* list, you can select steady-state or transient analysis type (time-independent or time-dependent).
- Under *Transient Thermal Data to Use...*, you can specify the use of transient boundary conditions in a steady-state problem.
- Under *Thermostat*, you can select the algorithm to use for evaluating sensor temperature if any thermostats are used.
- In the *Advanced* group, you can add extra solution control tools.
- In the *Parallel Processing* group, you can set up parallel solvers.

In the *Solution Units* tab, you can redefine units of measurement for the variables used in the problem. In the *Ambient Conditions* tab, you can specify ambient conditions. In the *Initial Conditions* tab, you can set up parameters of transient problems and specify the initial state of the system. In the *Restart* tab, you can set up the restart of a previously stopped simulation. The *Thermal* tab is available only for the Advanced Thermal module. This tab allows the addition of custom subprograms, specify compressibility settings for *1D Duct* elements, and accounting for ablation and charring effects. You can use the *Transient Setup* tab to set up parameters of transient processes. You can use the *Results Options* tab to create data samples that are fed into the results processing module after the simulation is complete.

After setting up the parameters of the problem (you can also make the settings later), click *OK* in the *Solution* dialog box to switch to the problem formulation mode. You can formulate the problem using the tools described in the next section.

5.3.2. Formulating the complex heat exchange problem

The **Advanced Simulation** toolbar contains a set of tools that you can use to prepare the model for numerical modelling. These operations include geometry processing, building a simulation mesh, specifying or redefining physical properties of materials, and setting up boundary conditions, thermal coupling, and so on (Figure 5.4). This section covers only tools relevant to problem formulation, namely *Simulation Object Type*, *Load Type*, and *Constraint Type*.

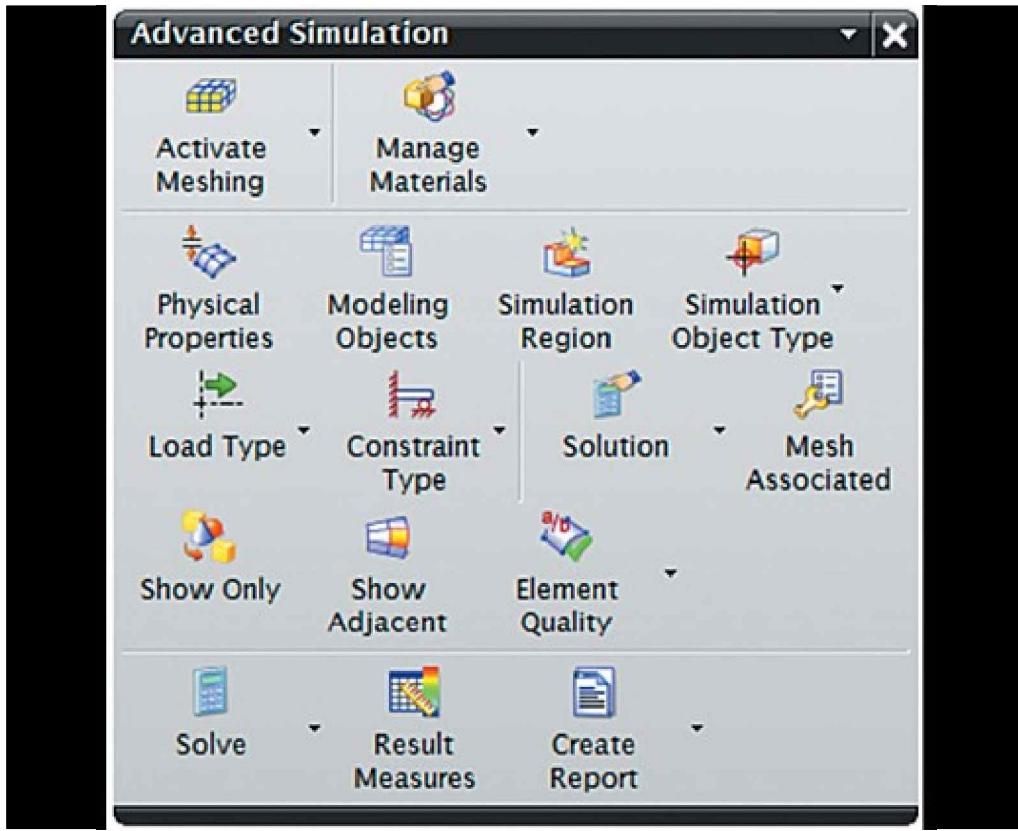


Figure 5.4. Advanced Simulation toolbar

You can use the **Simulation Object Type** tool set shown in Figure 5.5 to set up boundary conditions, create thermal couplings, and account for a variety of physical phenomena. The following is a detailed description of each tool.



Figure 5.5. Simulation Object Type tools

You can use the **Articulation** command to define motion (translation or rotation) of objects in the simulation model. This command is applicable to 2D, 1D, and 0D objects in an transient formulation only. Positions of all moving objects are calculated for each time increment. Recalculation intervals for angular coefficient (if radiant heat exchange is taken into account), temperature loads, and thermal coupling parameters are selected in accordance with the temporal discretization of the problem. The main parameter of the command is the law of motion of the object as defined using the **Modeling Objects** tool (Figure 5.6). You can define complex translational-rotational motion by creating two or more **Modeling Objects**. Translational motion is specified as a motion vector and a velocity. For rotational motion, you specify a rotation point, a rotation vector, and a speed of rotation.

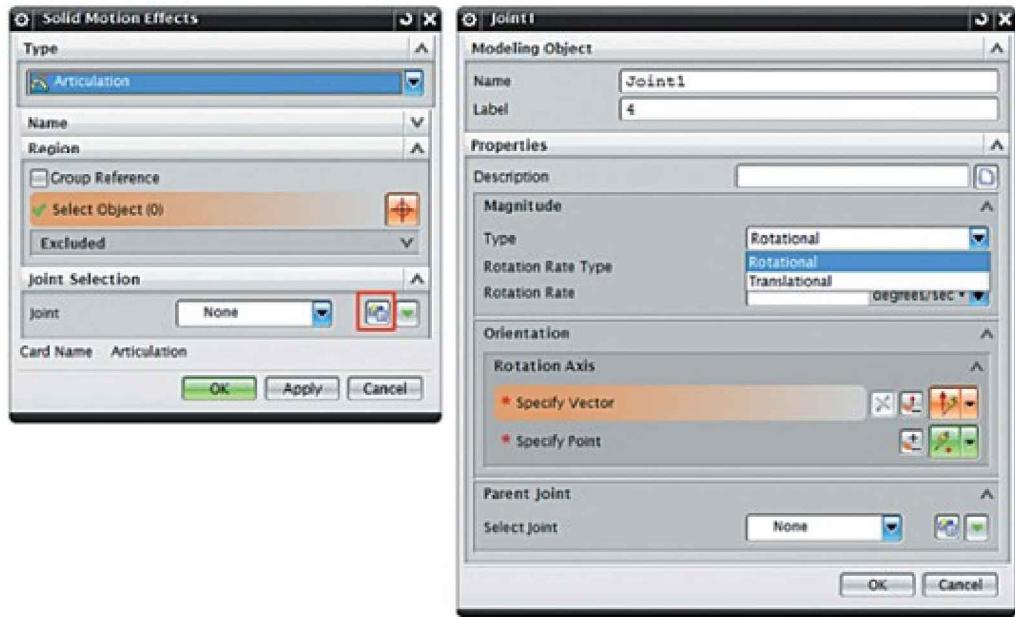


Figure 5.6. The Articulation tool

Deactivation Set/Deactivation Set Advanced. This tool excludes some of the simulation model elements from the simulation process. This command can be applied to individual elements and sets of 1D, 2D, and 3D elements. You can set up the parameters to deactivate in *Deactivation Options*. Select the *Eliminate from Solution* check box for the selected elements to exclude all calculation types. Select the *Deactivate Radiation* check box to exclude only radiant heat exchange processes.

You can use the **Duct Flow Boundary Conditions** tool set to model flow and heat exchange in ducts using duct-type 1D elements. You can apply the condition to 1D elements, curves, or edges. The direction of flow along the 1D element is specified when 1D elements are constructed (Figure 5.7, A). You can then visualize this direction using the direction of the X vector when you display the mesh collector (Figure 5.7, B).

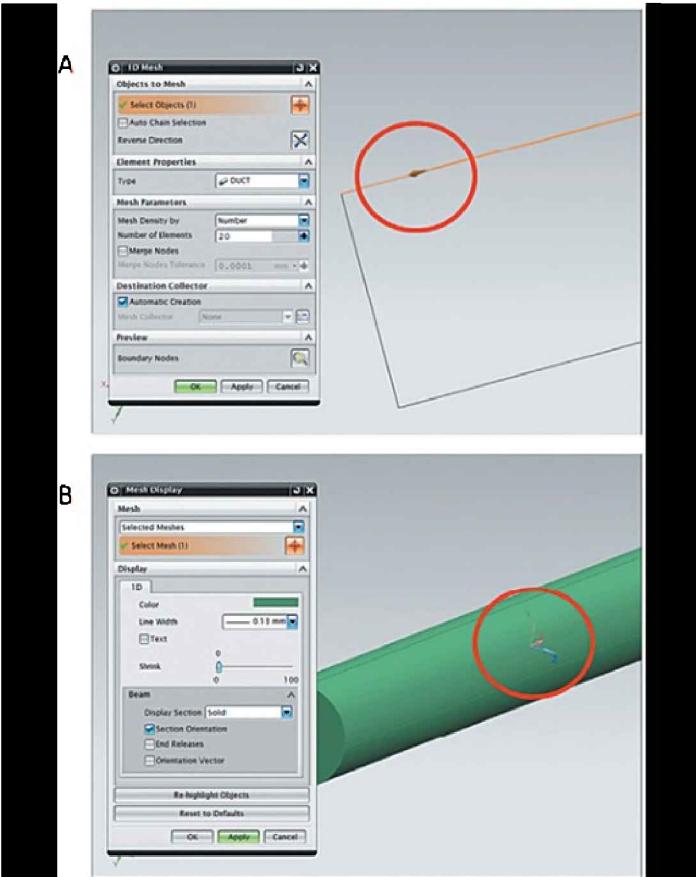


Figure 5.7. Specifying (A) and subsequently visualizing (B) the direction of flow along a 1D duct

You can use **Duct Flow Boundary Conditions** to specify the following parameters of flow in a 1D duct:

- **Duct Fan/Pump.** Specify (volume or mass) flow rate, velocity or pressure head created by a fan or a pump in the conduit. If necessary, you can reverse the flow direction in the duct. Note that you cannot reverse the direction by specifying a negative velocity or flow value. To reverse the direction, select the *Reverse Flow Direction* check box as shown in Figure 5.8, A.
- **Duct Opening.** Pressure and temperature values at the open boundary of the duct. This condition is automatically applied to all outer boundaries of conduit-type elements for which no boundary conditions are defined. By default, pressure and temperature values are taken from *Ambient Condition*. To specify different

values, in the *External Temperature* list and the *External Total Pressure – Gauge* list, select the desired value. You can enter pressure and temperature only as a constant value or a function of time. You can additionally specify head loss at the boundary. To specify head loss, use the **Create Modeling Object** tool (Figure 5.8, B). To make this option available, select the *Head Loss Control* check box. This tool is described in detail in para. 5.5;

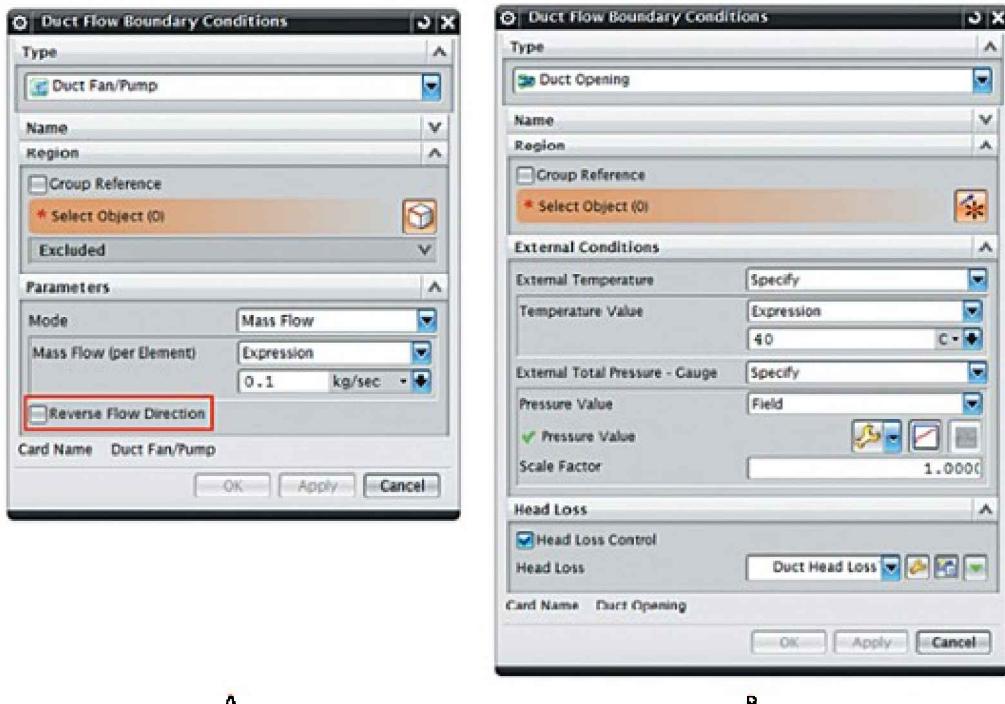


Figure 5.8. Specifying Duct Fan (A) and Duct Opening (B) boundary conditions

- **Duct Pressure.** Specify the total pressure value. The pressure is specified as gauge pressure. In case of transient formulation, you can specify pressure as a function of time.
- **Duct Flow Properties.** You can use this tool to specify surface roughness values and head losses. Depending on the selected options, you can calculate losses using roughness and geometry parameters (length, perimeter of internal cross-section, area of internal cross-section), or the specified head loss coefficients. This type of boundary conditions is covered in greater detail in para. 5.5.

You can use the **Interface Resistance** tool to specify additional conductance (resistance) between heat exchange objects. It can be specified as *Total Conductance*, *Total Resistance*, *Conductance per Length*, or *Heat Transfer Coefficient*. Depending on the type of coupled objects, you can use the following *Type* options: **Surface Interface**, **Edge Interface**, or **Interface Between Two Sets**. The type of coupling determines the available interaction objects: external surface faces of 3D elements, edges of 2D elements, or a combination of these.

The **Surface Interface** type lets you enter additional heat transfer parameters at the contact surface of bodies. If two identical materials are coupled at the level of FE meshes, the boundary between bodies is not used in calculations (Figure 5.9, B). If this boundary must not be neglected (for example, if there is additional resistance or conductance at the boundary), you can specify a **Surface Interface** to take this effect into account (Figure 5.9, A). You can specify heat transfer parameters at the contact boundary as total conductance, total resistance, or heat conductivity coefficient. If necessary, you can take into account the non-uniformity of heat transfer at the boundary. This non-uniformity is defined as a spatially distributed dimensionless multiplier for the base value.

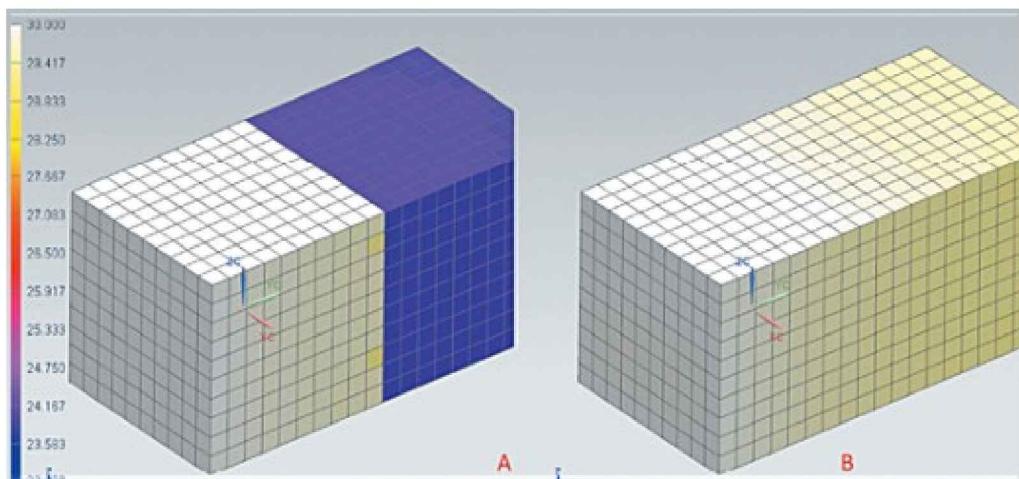


Figure 5.9. Temperature distribution in two coupled bodies with specified resistance on the surface of the contact (A) and without such resistance (B)

The **Edge Interface** type is similar, however instead of a surface, an edge is specified as the boundary. In addition, conductance per length is specified instead of heat conductivity coefficient.

If you need to create an interface between two surfaces or edges that form a T-juncture, use ***Interface Between Two Sets***. For this contact type, you need to select a *Primary Region* and a *Secondary Region*. The thermal coupling can be defined through total resistance, total conductance, conductance per length or heat transfer coefficient.

To account for heat emission due to electric current, use the ***Joule Heating*** tool. To describe this heat generation type, specify *Current*, *Voltage*, and *Electrical Coupling* parameters. Electrical coupling is the ohmic resistance on the part of the circuit where current and voltage parameters are defined. If you select *Electrical Coupling*, specify additional parameters: *Coupling Resolution* describes the coupling precision of *Primary* and *Secondary* regions. If necessary, also select the option of coupling only overlapping regions.

Override Set – Thermal Properties. You can use this tool to override thermal properties of model elements. This command applies properties specified in the override set to the simulation. Initially specified properties are not deleted. If you turn off or delete the ***Override Set — Thermal Properties*** simulation object, the system automatically uses the properties specified at the *FEM* level. You can use this command to override: thermal conductivity, orthotropic thermal conductivity, specific heat, electrical resistivity, emissivity factor, and solar absorptance. You can also override properties of laminate shells. The value of the overridden parameter can be uniform or variable in space.

To create simulation models for Peltier effect heat pumps, you can use the ***Peltier Cooler*** command. You can specify physical properties of the material, the number of *p-n* junctions, voltage or current, position of hot and cold sides, the form factor, and the Seebeck coefficient.

You can use the ***Radiation*** command to select objects that participate in radiant heat exchange. If you use this tool, radiant heat exchange is set up either for all objects that have thermo-optical properties specified (*All Radiation*), or manually selected objects only (*Enclosure Radiation*). Only the selected part of the surface radiates (the top and/or bottom surface). The available settings include the angular coefficient calculation algorithm, the precision of angular coefficient calculation, the additional element subdivision coefficient, whether the ambient temperature is taken into

account, and selection of elements for shadow zone calculation. If you create a limited set of radiating surfaces, the top and bottom sides of the radiating objects have to be selected individually. The radiating side is determined in accordance with the direction of the surface normals. To display and change the normal direction if necessary, switch to the FEM file and use the ***Finite Element Model Check*** command. When you work with the **SIM** simulation model, you can only visualize normal directions.

The ***Radiative Element Subdivision*** simulation object allows local redefinition of the *Element Subdivision* option for ***Radiation***, ***Radiative Heating***, and ***Solar Heating*** commands. In fact, this object is used to override element subdivision parameters. It can increase the precision of subdivision (which results in improved calculation precision as well as longer calculation time) or reduce the precision to speed up the calculation.

You can introduce a source that radiates in the infrared, visible, or user-defined spectrum using the ***Radiative Heating*** command. If you select a collimated beam, you need to specify its incidence vector. The main parameter you need to set up is the amount of radiant energy (*Power Value*), which can be specified as total power or as per-element radiation power, or as radiant flux. All objects with thermo-optical properties defined can be set up as radiators. Alternatively, you can use only user-selected objects (which must still have thermo-optical properties defined). Radiation is specified separately for each side of the radiating surface. You can also separately specify the radiation *Spectrum*: visible, infrared, or user-defined spectral bands. The *Calculation Method* is another option you can set up. There are two calculation methods available: *Calculation Method-Deterministic* and *Monte Carlo*. If you select the deterministic method, you need to specify the *Element Subdivision* parameter, which influences the precision of angular coefficient matrix calculation: the finer the subdivision, the more precise the calculation (*Very Precise*). If you select the Monte Carlo calculation method, you need to specify parameters of radiating elements: *Ray Density Control* as shown in Figure 5.10. If you select *Ray per Element* or *Ray per Area*, you also need to specify the number of emitted rays. The number of rays must be equal to or greater than the number of elements participating in radiant heat exchange. If you select the *Based on Error Criterion* option, specify statistical parameters of the radiation source, such as *Error Criterion*, *Confidence Level*, and *Random*.

Number Generator. By default, the Monte Carlo method is used to calculate angular coefficients as well as to calculate heat fluxes for radiant heat exchange. If you want to use the Monte Carlo method only for angular coefficients, you need to select the *Calculation View Factors only* check box. In this case the rest of parameters are calculated deterministically.

In addition to surfaces, faces of solid-body elements can participate in radiant heat exchange. To activate them in the calculation, select the *Include Faces of Solid Elements* check box. In the *Distribution* group, you can specify the method of spreading the load across the radiating element. The distribution can be *Uniform* or non-uniform (*Spatial*).

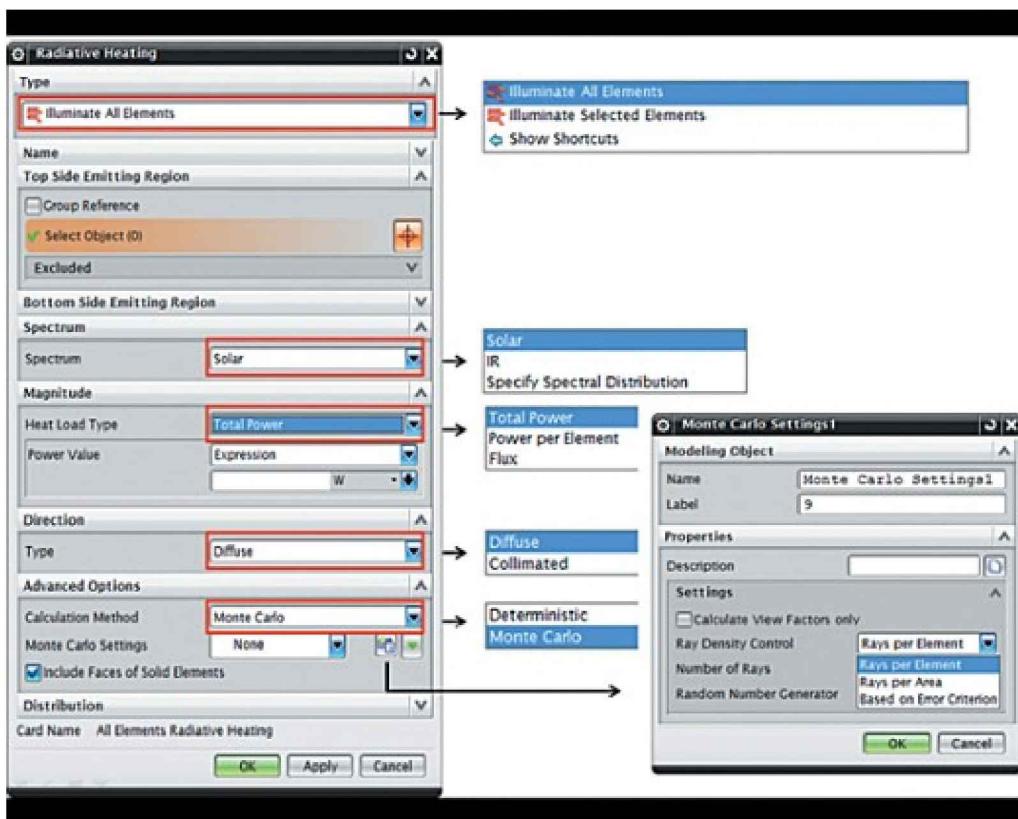


Figure 5.10. Setting up radiant heat exchange calculation using the Monte Carlo method

You can use the **Report** tool to view local and integral values of some parameters in numerical form immediately when the simulation finishes (Figure 5.11). If you need to track the temperature value on selected objects during simulation, in the *Type* group, use the *Track During Solve* option. The

only parameter of this report is *Number of Iterations between Outputs*, which specifies the intervals for displaying the relevant data.



Figure 5.11. Data sets for reporting

You can view summary heat exchange information and values of angular coefficients for a given object in the *Type* group list using the *Heat Maps* report. This report contains all heat exchange data in a summary table as shown in Figure 5.12. It also contains angular coefficient values as shown in Figure 5.13.

Heat Maps

To(j)	From(j)	Time	Temp(i)	Temp(j)	Conduction	Radiation	Convection	Linear Thermal Coupling	Total
Report(1)	Report(3)	0	100	29.1454	NA	-802729	NA	NA	-802729
Report(1)	Radiative Environment	0	100	29	NA	-9.07667e+006	NA	NA	-9.07667e+006
Report(1)	Rest	0	100	29	NA	NA	NA	NA	NA
Report(1)	All	0	100	NA	0	-9.8734e+008	0	0	-9.8734e+008
Report(3)	Report(1)	0	29.1454	100	NA	802729	NA	NA	802729
Report(3)	Radiative Environment	0	29.1454	29	NA	-802795	NA	NA	-802795
Report(3)	Rest	0	29.1454	29	NA	NA	NA	NA	NA
Report(3)	All	0	29.1454	NA	0	-65.3125	0	0	-65.3125
Radiative Environment	Report(1)	0	20	100	NA	9.07668e+006	NA	NA	9.07668e+006
Radiative Environment	Report(3)	0	20	29.1454	NA	802785	NA	NA	802785
Radiative Environment	Rest	0	20	29	NA	NA	NA	NA	NA
Radiative Environment	All	0	20	NA	0	9.87346e+008	0	0	9.87346e+008

Figure 5.12. Summary heat exchange table for the two selected regions and the environment

View factors

From[i]	To[j]	Time	Emissivity (i)	Emissivity (j)	Area[i]	Area[j]	Black Body View Factor(i,j)	Gray Body View Factor(j,i)	ScriptF (i)	RADK(j,i)
Report[1]	Report[3]	0	0.8	0.8	20000	20000	0.100067	0.0877918	0.0782335	1404.67
Report[1]	Radiative Environment	0	0.8	1	20000	4.84e+012	0.799903	0.912207	0.729706	14595.3
Report[1]	Rest	0	0.8	0.8	20000	10000	0	N/A	N/A	N/A
Report[3]	Radiative Environment	0	0.8	1	20000	4.84e+012	0.89997	0.919556	0.735645	14712.9
Report[3]	Rest	0	0.8	0.8	20000	10000	0	N/A	N/A	N/A
Radiative Environment	Rest	0	1	0.8	4.84e+012	10000	2.99919e+009	0.181832	0.181832	8.80067e+011

Figure 5.13. Values of angular coefficients for two selected regions and the environment

Depending on the selected report type, the results are stored in different files. Information about these files is shown in Figure 5.14.

Report Types	Data Types	File Names
Per Element	Text	REF
Per Region	HTML and text (.csv)	[simulation name]-[solution name].GroupReport.htm GroupReport.csv
Between Regions	HTML and text (.csv)	[simulation name]-[solution name].GroupReport.htm GroupReport.csv
Track During Solve	Text (.csv) and graph (.png)	TrackReportThermal.csv [report name] Solid Temperature.png
Heat Maps	HTML and text (.csv)	[simulation name]-[solution name].GroupReport.htm GroupReport.csv

Figure 5.14. Data storage formats for the Report tool

You can create a model that describes solar radiation using the **Solar Heating** command. To describe the model correctly, you need to specify: the spatial position and orientation of the body, the duration of insolation, the calendar date, and the influence of various extrinsic factors. All objects or user-specified objects can be subjected to this type of loading. These objects need to have thermo-optical properties specified.

Spatial orientation (Figure 5.15) is specified in the *Parameters* group and the *Model Orientation* tab:

- The *Orientation Method* parameter determines the way you describe the position of the object relative to the Earth: you can use either *Latitude*, that is, specify the position of the object using geographical latitude, or alternatively you can use *Sun Earth Vectors*, that is, specify the way the relative orientation of the Earth and the Sun changes with time.

- As the *Calculation Method* parameter, you can select either the deterministic method or the Monte Carlo method.
- If you select the *Latitude* orientation method, specify the orientation of the body with respect to the planet in the *Model Orientation* tab. Orientation is specified as two vectors: *Aim* and *Align with*. The *Day Segment* is the time interval for which the simulation is carried out. The *Number of Equal Angular Intervals* is the number of points on the trajectory of the Sun for which the calculation is carried out.

If, as the *Day Segment*, you specify *Partial Day*, you need to set up the following additional parameters:

- *Start Position* is the starting point of the Sun in the simulation.
- *Final Position* is the end position of the Sun.
- *Analysis Time at Start Position* is the moment in time that is used as the starting point for calculating solar heat loading. This time must be less than the end time of the simulation.

If, as the *Day Segment*, you specify *Single Position*, you only need to specify the single moment in time for which the analysis is carried out.

The *Solar Flux* tab (Figure 5.15) is used to specify solar irradiance parameters. The source data include the *Time of Year* and *Account for Atmospheric and Other Earth Effects*. When you select the *Account for Atmospheric and Other Earth Effects* check box, the additional *Atmospheric and Other Earth Parameters* group becomes available. Here you can specify additional factors such as *Apparent Solar Radiation* or *Atmospheric Extinction Coefficient*. These factors are used together to calculate direct solar irradiance and to define the attenuation factor. The default value corresponds to direct irradiance at sea level on a clear day.

- You can use the *Clearness Factor* to set up irradiance relative to sea-level irradiance at the given location.
- The *Diffuse Sky Radiation Factor* is an additional correction for scattering of radiation by the atmosphere.
- The *Ground Surface Reflectance (Albedo)* defines the albedo.

If in the *Solar Heating* dialog box, in the *Type* group you select the

Illuminate Selected Elements radiation type option, you need to specify objects that are hit with solar energy. These objects are subdivided into “top” and “bottom”-irradiated.

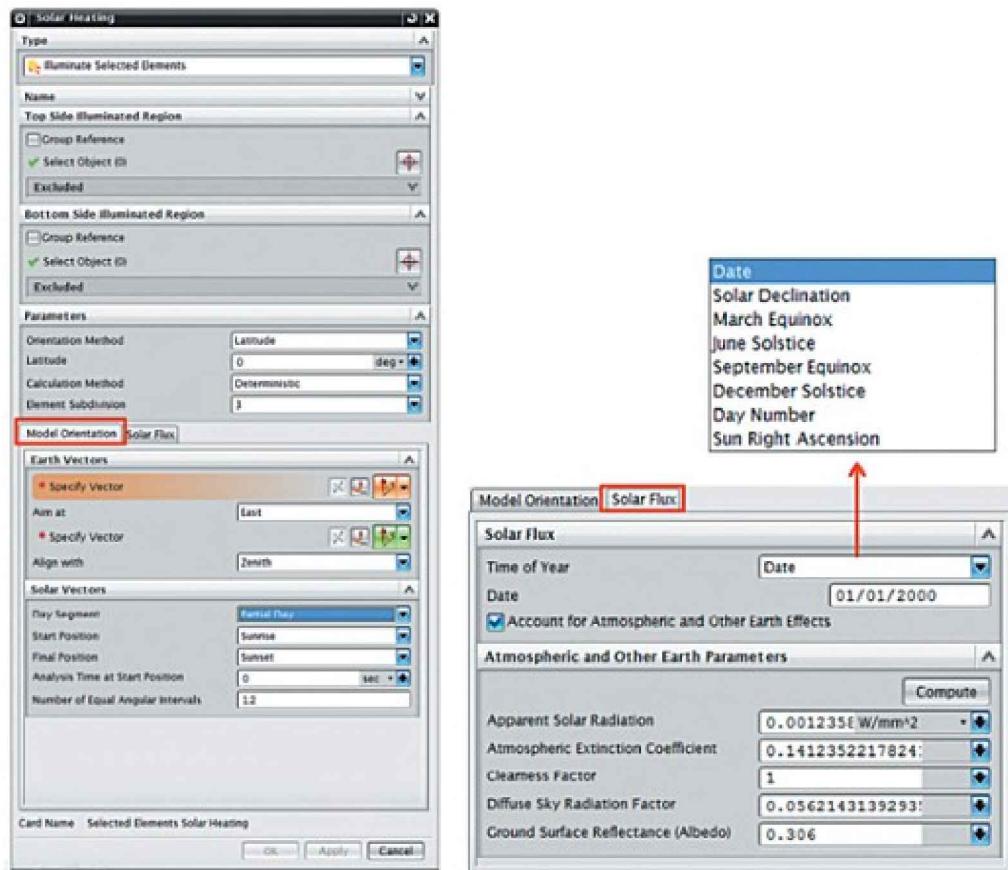


Figure 5.15. Solar Heating dialog box

You can model heat exchange between two interacting surfaces of different objects using the **Surface-to-Surface Contact** command. The surfaces that exchange heat can be in contact or be separated by a gap. You can select the interacting surfaces manually or automatically as shown in Figure 5.16 (a detailed description is provided in Chapter 4 of Part 1).

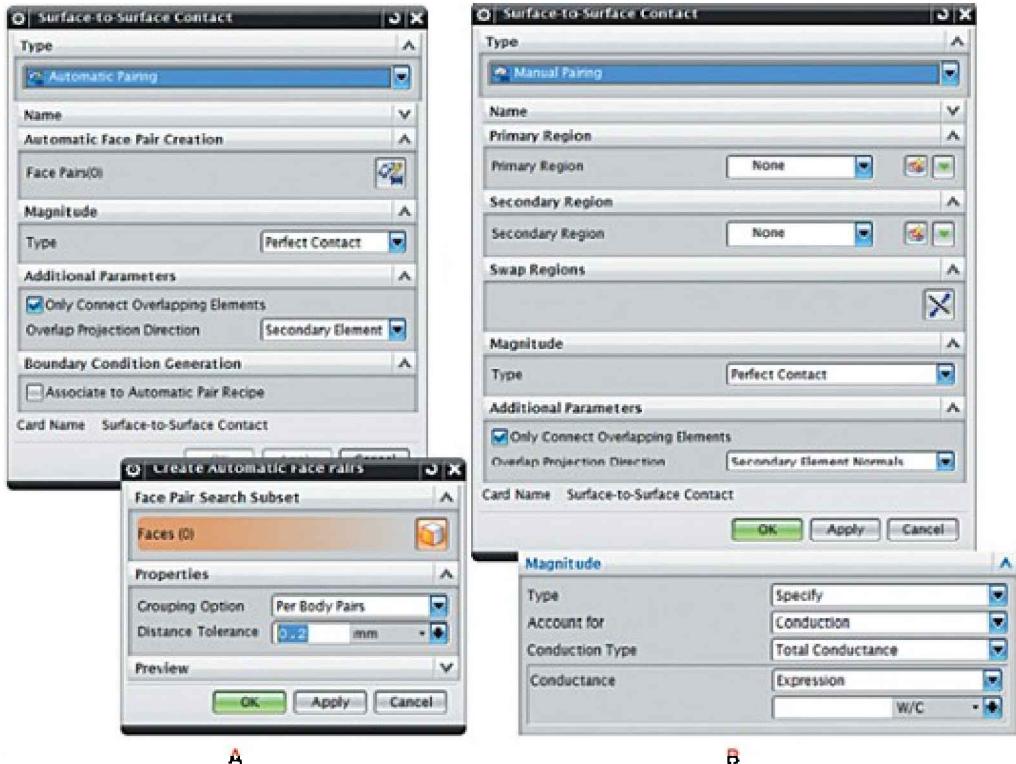


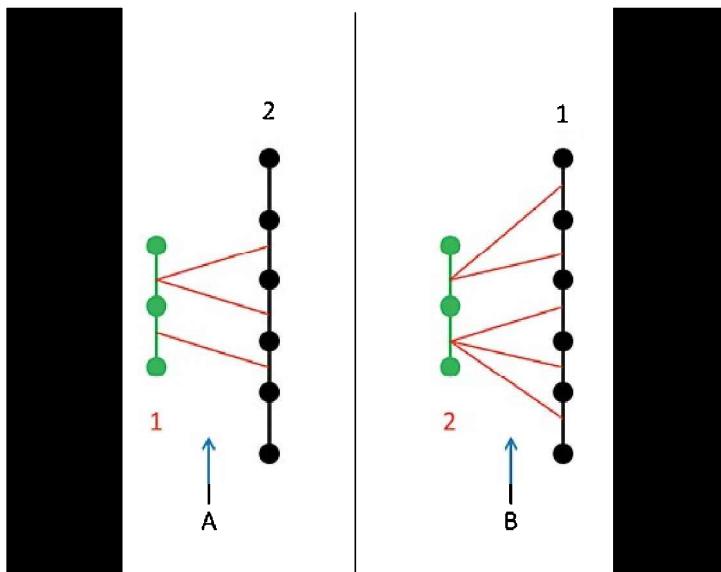
Figure 5.16. Tools for creating coupling with the Surface-to-surface contact type in automatic (A) and manual (B) modes

To create a contact pair, open the **Surface-to-Surface Contact** dialog box and set up the following parameters:

- Select the **Type** to create the contact with specific properties: **Specify** or ideal contact interaction (**Perfect Contact**). If you select perfect contact, the surface-to-surface coupling will not impede heat flow. If you select **Specify**, you need to describe the heat transfer mechanism: **Conduction** and/or **Radiation**. Specifying a radiation-type thermal coupling is effective if angular coefficients are known or if the objects interact without radiant energy losses. If such data is not available, the radiant heat exchange is modelled via **Radiation**. In cases where both radiation and conduction are present, both parameters are specified.
- Under **Coupling Resolution**, you can set up the precision of coupling element pairs and define the degree of additional virtual subdivision applied to each element.
- Select the **Only Connect Overlapping Elements** check box to

enable heat exchange only for element pairs that project normally onto each other. Specify the object (primary or secondary) on whose elements the normal originates.

If you specify the object as a *Primary Region* and *Secondary Region*, you need to observe several rules related to the method of representing these regions and subsequent processing in the simulation process. The primary region differs from the secondary region in that each surface finite element on the primary region must be connected with at least one secondary finite element (the element on the secondary region). Due to this rule, the selection of the primary and secondary regions is typically based on the geometrical dimensions of the interacting regions. Except some individual cases, the smaller region is designated as primary, and the larger region is designated as secondary. Figure 5.17 shows correct and incorrect region selection. It is clear that if regions are selected incorrectly, superfluous thermal connections occur that cause errors. The primary and secondary elements are coupled by virtually connecting their centres of mass.



*Figure 5.17. Primary and secondary region selection principles: A. Correct.
B. Incorrect*

The primary and secondary regions differ in another important aspect: the additional subdivision of each primary finite element into subelements in accordance with the *Coupling Resolution* option. This virtual subdivision (the user cannot see the subdivision results on the FE mesh) allows more

precise connection of primary and secondary finite elements without creating a finer mesh. Here, not elemental centers of mass but subelemental centers of mass are connected. The greater the size difference between finite elements of the primary and the secondary region, the higher you should set the order of the *Coupling Resolution* parameter. For example, if the elements are of identical size, you can use the *One-to-One* connection. If you set up this option incorrectly, instances of local heat transfer can occur (Figure 5.18), because the *One-to-One* connection implies creating a single connection irrespective of the size difference of the elements.

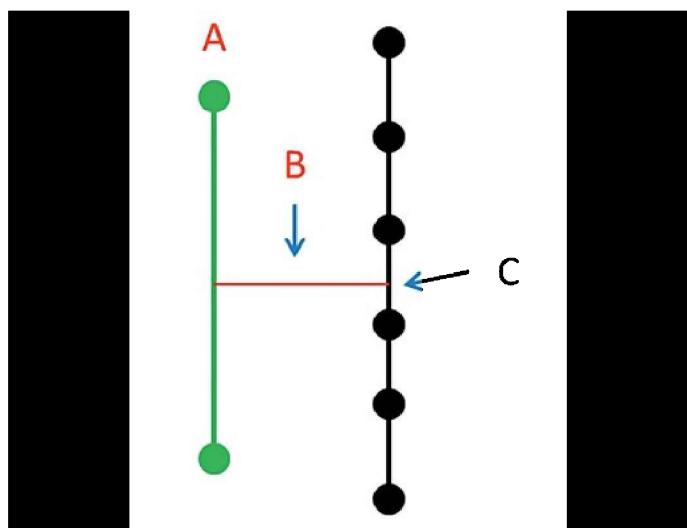


Figure 5.18. Incorrect thermal coupling creation (B) between a large primary FE (A) with local heat transfer in one secondary FE (C)

To localize the primary and secondary region interaction zone more precisely, you can select the *Only Connect Overlapping Elements* check box. This option allows using subelements instead of elements as the region interaction boundary. Figure 5.19 shows the results of analysis using overlapping elements. If you turn on this option for the primary and secondary regions that are not parallel, you also need to specify on which of the surfaces the normal will originate to determine the subelement overlap zone. Figure 5.20 shows possible localizations of this zone depending on the normal direction.

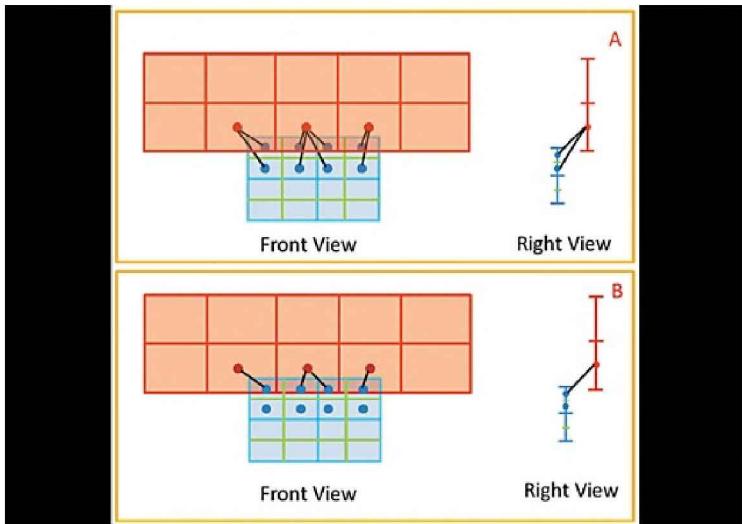


Figure 5.19. Creating thermal connections without element overlap (A) and with element overlap (B)

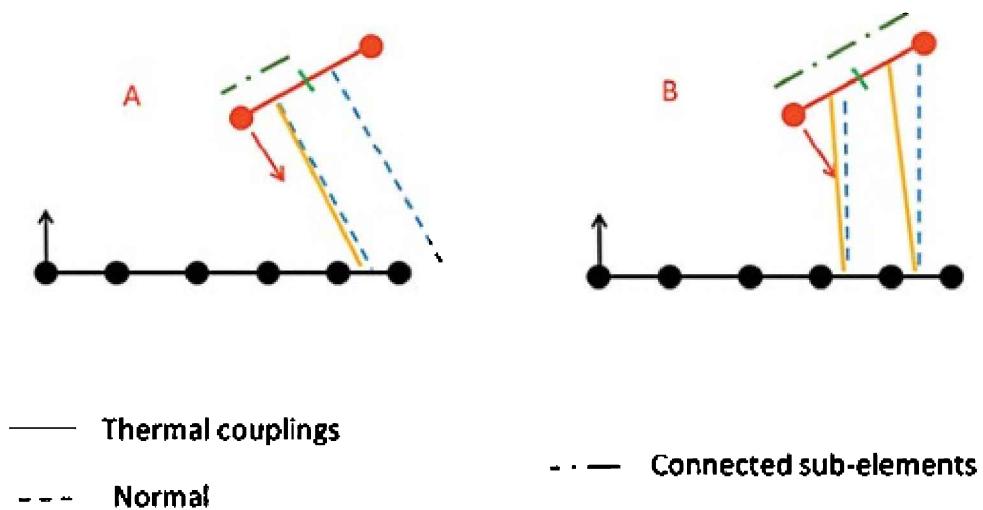


Figure 5.20. Subelements participating in the thermal coupling depending on the normal construction method. A. Normal to the primary region. B. Normal to the secondary region

To create heat conduction between contacting objects, you can use the **Thermal Coupling** tool. Unlike the **Surface-to-Surface Contact** connection, you can use this command to connect surfaces as well as 0D, 1D, 2D, and 3D elements in various combinations. Typically, this connection type is used to model contacting objects. If for whatever reason you need to use this tool to model heat transfer over a gap (between objects that do not

physically touch), you can use the *Conductive Gap* type, which can describe this process. You can set up the following options of the thermal coupling type for coupled objects:

- If you select *Total Conductance*, you need to specify the total conductance coefficient G (W/K).
- If you select *Total Resistance*, specify the inverse value of the total conductance (K/W).
- The *Heat Transfer Coefficient* (W/(m²K)) specifies the conductance between the primary and the secondary regions in accordance with the following expression: $G = A \cdot h$, where A is the area of contact.
- *Edge Contact* specifies conductance between the primary and the secondary regions in accordance with the following equation:
$$G = C_L \cdot L$$
, where L is the length of the edge and C_L is conductance per length (W/(mK)).

Conductive Gap specifies the conductance between the primary and the secondary regions in accordance with the following equation:
$$G = (k \cdot A) / L$$
, where A is the area of the contacting surfaces, L is the distance between the primary and the secondary elements (gap width), and k (W/(mK)) is the thermal conductance of the gap.

You can select objects for which the thermal coupling is created in the *Primary Region*, and *Secondary Region* groups using the *Select Object* tool.

You can specify the following additional parameters:

- Under *Coupling Resolution*, you can set up the precision of coupling element pairs and define the degree of additional virtual subdivision applied to each element.
- Select the *Only Connect Overlapping Elements* check box to enable heat exchange only for element pairs that project normally onto each other. Specify the object (primary or secondary) on whose elements the normal originates.

If the selected heat transfer type is non-uniformly distributed in space, you can set it up in the *Distribution* group. You can select the *Per Element* check box to set up heat transfer parameters for each of the elements instead of

using the surfaces. This option should be applied only to the *Total Conductance* type if finite element size is the same on the whole surface. Otherwise the distribution of conductance will be uneven.

The **Thermal Coupling – Advanced** tool contains additional options for creating thermal couplings. You can use this tool to create *Perfect Contact*, *1-way*, and *User Function*:

- *Perfect Contact* is a thermal coupling between the primary and the closest secondary finite element with virtually infinite conductance. This option should be used only if the meshes are mated or if the temperature gradient in the contact plane is insignificant.
- *1-way Conductance* is used when you need to create an unidirectional conductance: from the primary region to the secondary. You can specify either *Total Conductance* as defined by the total conductance coefficient G (W/K), or the *Heat Transfer Coefficient* (W/(m²K)) that sets the conductance between the primary and the secondary regions in accordance with the following expression: $eq_t_1.tif$, where A is the area of contact.
- You can use the *User Function* type to create custom heat conductance variants that can use custom procedures.

To set up a thermal coupling that imitates convective heat exchange, use the **Thermal Coupling – Convection** command. In the list of the *Type* group, you can select one of the four convective coupling types:

- *Convection Coupling* is used to model a convective thermal coupling of the fluid and the adjacent wall. The heat transfer rate is defined by the heat transfer coefficient that can be additionally corrected with *Duct Convection Correction*. The correction equation for the modified coefficient is of the form:

$$h = h_0 \cdot C \cdot ((T_{wall} + T_{fluid}) / 2T_{ref})^r \cdot |T_{wall} - T_{fluid}|^d,$$

where h_0 is the original heat transfer coefficient, C is the proportionality factor, r is the temperature ratio exponent, d is the temperature difference exponent, and T_{ref} is the reference temperature. The fluid itself is defined using *duct*-type 1D elements. You specify physical properties of the fluid and its flow rate for the 1D elements. The *Additional Parameters* group contains

parameters that are analogous to additional parameters of the ***Thermal Coupling*** command.

- You can model heat exchange between an object and the fluid that flows around it using the ***Forced Convection Coupling*** command. The motion of the fluid in this case is due to external influence, which is forced. For this heat exchange type, you need to specify the fluid flow regime near the wall. You can select ***Duct Developing Flow***, ***Duct Fully Developed Flow***, or ***Flat Plate in Free Stream***. Depending on the selected flow type, correlations corresponding to the flow regime are automatically applied. You can additionally specify a ***Multiplier*** that takes into account various aspects of the flow: the shape, spatial non-uniformity, time dependence, and so on. Additional parameters can be specified in a way similar to the ***Thermal Coupling*** command.
- ***Free Convection Coupling*** corresponds to free convective flow in the vicinity of standard objects. The standard objects can include: ***Horizontal Plate***, ***Inclined Plate***, ***Sphere***, ***Cylinder***, ***Parallel Plate Channel***. Calculation of heat transfer takes into account the direction of the gravity force specified in ***Ambient Conditions*** settings. In the ***Convection from*** list, specify which of the object's surfaces participates in the convection process: ***Top***, ***Bottom***, or ***Top and Bottom***. Depending on the selection of sides participating in convective heat exchange, the corresponding correlation dependencies are applied, which take into account the shape and orientation of the object. You can use the ***Characteristic Information*** option to specify the characteristic geometry dimensions that are used to calculate convective flows. If you use the ***Calculate from Element Set*** option, the initial information comprises shape and orientation data for the elements on the selected convective heat exchange surface. If you need to specify a characteristic linear scale along the flow direction and the direction of the normal vector to the surface, use the ***Specify*** option.
- ***Across Gap Convection Coupling***. You can use this coupling type to account for free convection that arises in gaps between horizontal parallel plates (***Between Large Parallel Horizontal Plates***), or inclined parallel plates (***Between Large Parallel Inclined Plates***).

Plates). Heat transfer between concentric spheres can be modelled using the *Between Concentric Spheres* type. If you need to study the flow in coaxial horizontal cylinders, you can use the *Between Long Concentric Horizontal Cylinders* type. To specify characteristic dimensions and distances between interacting shells, you can use automatic calculation (*Calculate from Element Set*) or specify the values manually (*Specify*). In the latter case, you need to specify the characteristic dimension, the distance between surfaces, and the normal to surfaces that participate in heat exchange. The fluid that fills the gap between the interacting surfaces can be specified in the *Environment* group.

To account for the radiative component of heat exchange, use the ***Thermal Coupling – Radiation*** command and set up the following parameters:

- The *Type* of the thermal coupling. If the positions of emitting objects allow setting the angular coefficient to one, select the *Gap Radiation* type. If angular coefficients are not equal to one, select the *Object to Object Radiation* type. In this case two unidirectional connections are created between the primary and the secondary surfaces. The coupling itself is not created between each primary and secondary element. Instead, it is created between the primary (secondary) element and the average temperature of the set of secondary (primary) elements (similarly to emission into the ambient medium with a given temperature).
- You can use the *Magnitude* option to specify the absolute value of the radiation. In the *Type* list, you can select one of three methods: *Close Parallel plates*, *Gray Body View Factor*, or *Effective Emissivity*. The radiation *Magnitude* itself is calculated using the following formula:

$$\text{Magnitude} = \sigma \cdot GBVF \cdot \varepsilon_1 \cdot A_1 (T_1^2 + T_2^2)(T_1 + T_2),$$

where σ is the Stefan-Boltzmann constant, *GBVF* is the *Gray Body View Factor* value you specified, ε is the emissivity factor of the primary region, *A* is the area of overlap of the primary and the secondary regions, T_1 is the absolute value of the primary region temperature, and T_2 is the absolute value of the secondary region temperature.

The type is usually chosen depending on the problem at hand. For example, *Gap Radiation* is typically used for problems such as radiation between plates of multi-layer heat insulation, honeycomb panels, and other cases where pairs of surfaces are separated by a narrow gap and have identical or similar geometrical dimensions. For this type, the *Close Parallel plates* radiation magnitude type implies using the $GBVF = 1/(1 + \varepsilon_1/\varepsilon_2 - \varepsilon_1)$ relation, where ε_1 , ε_2 are emissivity factors of the primary and the secondary regions, and *GBVF* is the gray body view factor. If you select *Gray Body View Factor* magnitude type, this parameter is not calculated but instead provided by the user. It is notable that in this method, the emissivity factor is always specified for the *Top* surface, which is particularly important if using 2D elements. Similarly, if you select *Effective Emissivity*, you need to specify the value of the complex defined as the product of *GBVF* and the emissivity factor of the primary region. In this case the emissivity factors specified in thermo-optical properties of the primary region are ignored.

5.3.3. Specifying thermal loads

You can set up heat sources with the **Thermal Loads** command on the **Advanced Simulation** toolbar. The sources can be specified as (Figure 5.21):

- *Heat Load*, that is, the power of the source (W).
- *Heat Flux* (W/m²).
- *Heat Generation* of a volume heat source (W/m³).

Each of the thermal loads can vary with time or in space, you can manage them with a thermal controller you created or apply them to multi-layer objects.

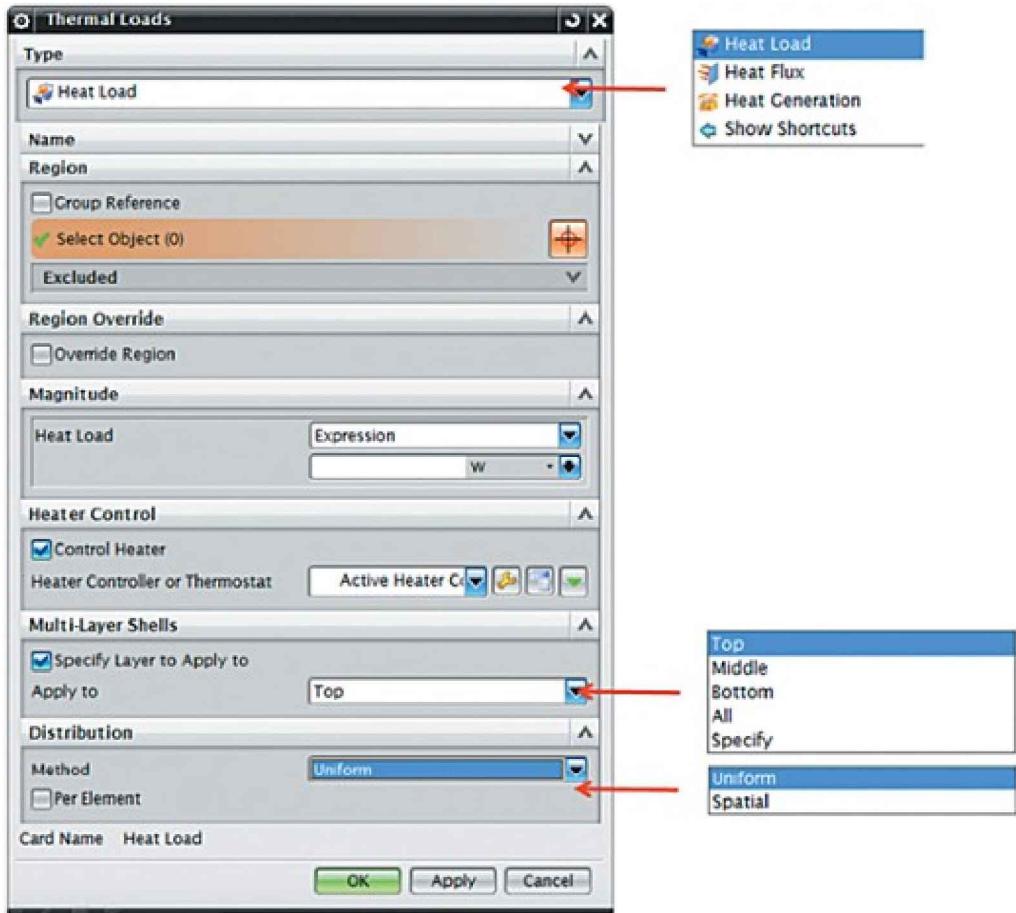


Figure 5.21. Thermal loads

Multi-layer shells are described in section 5.2.2.

5.3.4. Specifying boundary and initial conditions

Another tool that can be used in formulating the problem and setting up boundary/initial conditions is the **Constraint Type** command of the **Advanced Simulation** toolbar (Figure 5.22).

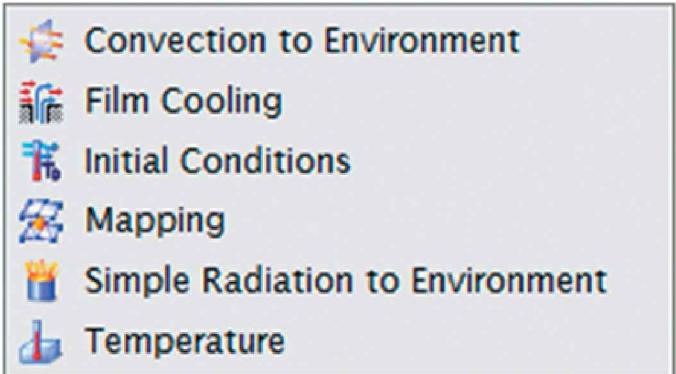


Figure 5.22. Constraint Type tool

You can use the **Convection to Environment** command to specify the constraint that determines the free or forced convection value in cases where the convective heat exchange coefficient value is known, or the object has a simple shape (Figure 5.23). To use this command, specify the *Type* of convective heat exchange:

- The **Convection to Environment** type allows describing the convective heat exchange process in the *Magnitude – Convection Coefficient* group if the convective heat transfer coefficient is known. The coefficient can vary with time or be spatially non-uniform (*Convection Coefficient – Spatial*). Moreover, you can specify this coefficient as an exponential function with a given convective parameter (*Parameter and Exponent*). The ambient *Temperature* should also be specified in the *Environment* group.
- You can use the **Free Convection to Environment** type to specify free convection cooling regime. You can use this condition, which includes empirical relations, to calculate flow near *Inclined Plates* and *Horizontal Plates*, *Sphere* and *Cylinder* objects, as well as flow in channels formed by parallel plates (*Parallel Plate Channel*). The calculation takes into consideration the orientation of the object with regard to the gravity vector. Convective fluxes from the top and the bottom sides of the plates (with regard to the normal to the surface) are considered separately. Shape and size of the plates is accounted for using *Characteristic Information*. You can use the *Multiplier* to correct the resulting coefficient and account for shape, size, and other factors that influence the efficiency of convective heat transfer. The ambient conditions are specified in

the *Environment* group using the *Fluid Material* and *Temperature* options.

- You can use *Forced Convection to Environment* to describe forced convection for a set of objects. These objects include: *Plate Aligned with Free Stream*, *Sphere in Flow*, and *Cylinder across Flow*. For all objects, you need to specify the characteristic linear dimension, the fluid of the incident flow, and its velocity and temperature. For a plate, you can additionally specify the side that participates in convective heat exchange.

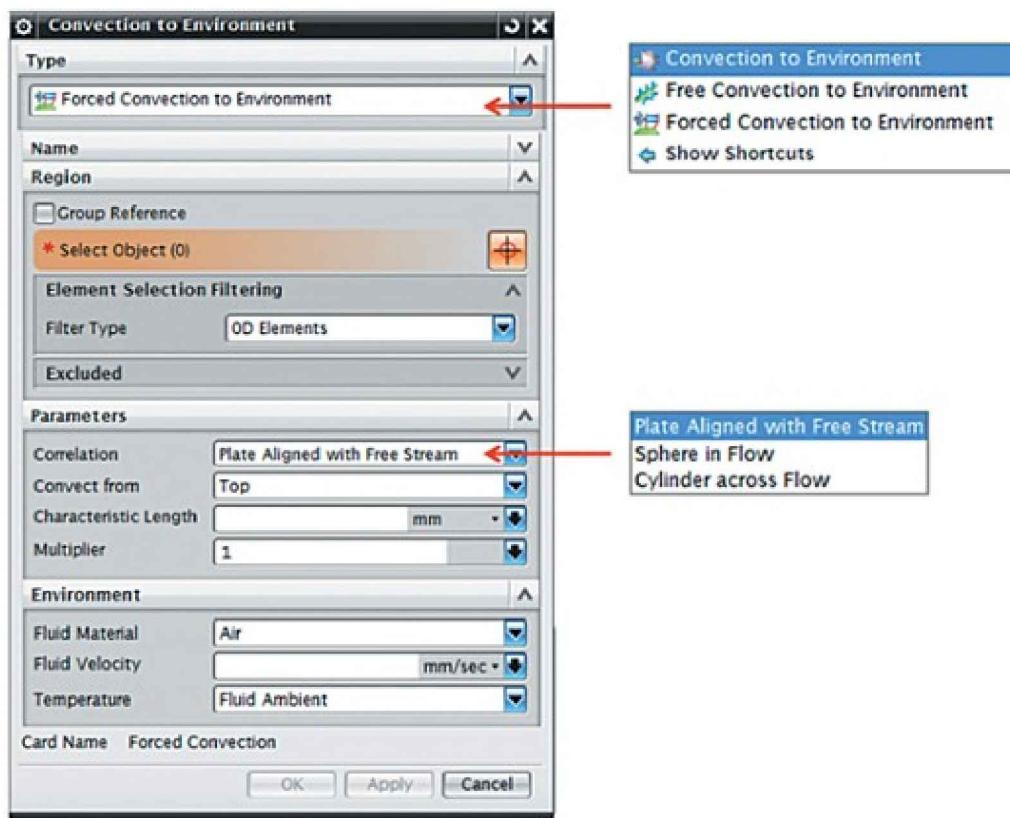


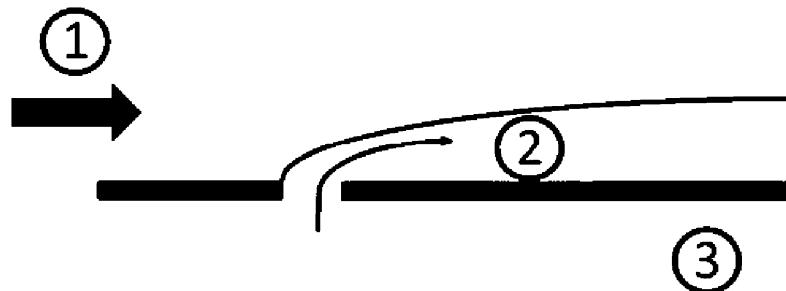
Figure 5.23. Convection to Environment command parameters



Figure 5.24. Film Cooling command parameters

You can use the **Film Cooling** command to specify an implicit model of surface cooling with a film of liquid without directly modeling the interaction of incident flow, the liquid film, and the cooled body. To specify film cooling, do the following (Figure 5.24): specify the region of surface where cooling takes place (1), select an edge, curve, or 1D element to specify the region where the cooling film is located (2), specify the magnitude of convective heat exchange (3) as a constant value or a function of time. The coefficient

can be spatially non-uniform (4). The efficiency of film cooling is adjusted using the corresponding parameter (5), which can be constant or a function of time and spatial coordinates (6). The temperature of external flow is specified in (7).



- (1) Incident flow temperature T_{fs}
- (2) Temperature of the liquid film T_f
- (3) Temperature of the wall T_w

Figure 5.25. Film cooling model setup schematic

The film cooling condition is similar in principle to the convection to environment condition, however, the adiabatic wall temperature is calculated from the incident flow temperature T_{fs} (1) and the temperature of the liquid film T_f (2) (Figure 5.25). The heat transfer due to convection is calculated as

$$Q = hA(\eta T_f + (1 - \eta)T_{fs} - T_w),$$

where h is the specified convection coefficient, A is the area of the selected convective heat exchange region, η is the specified convective heat exchange efficiency, T_w is the temperature of the wall.

You can specify the initial temperature distribution in the object using the **Initial Conditions** command.

The **Mapping** command defines the area of the model that contains results for use as the loading conditions for a coupled problem. To use this command, select display **Type**:

- You can use the *Thermal Zone* type to define a set of source model elements in which the temperature values need to be recorded. This option allows transferring data to layers of multi-layer objects.
- You can use the *Transverse Gradient Pair* type to define corresponding pairs of top and bottom boundaries of the source and target models to create a temperature gradient in the target model.
- You can use the *Exclude Elements* type to create a set of source model elements whose data must not be transferred to the target model.

You can use the ***Simple Radiation to Environment*** command to account for radiant heat exchange involving one or several surfaces with known emissivity factors and the environment with a known ambient temperature. As initial data, you can set up either *Gray Body View Factor – GBVF*, or *Effective Emissivity*, in which case you specify an emissivity factor and *GBVF*.

In this case, the heat flux can be calculated using the following equation:

$$q_r = \sigma VF \varepsilon_s A_s (T_s^4 - T_e^4)$$
, where ε_s is the emissivity factor of the surface, A_s is the area of the surface, T_s is the temperature of the surface, and T_e is the ambient temperature. You can set the ambient temperature for *Environment*, *Fluid Ambient*, and *Radiative Ambient* groups in the ***Ambient Condition*** tab of the ***Solution*** dialog box. To specify ambient temperature directly in the *Temperature* list in this command, use the *Specify* option.

You can specify temperature as a boundary condition for the selected objects using the ***Temperature*** command. You can enter temperature as a specific value by selecting *Option* list item *Specify*, by using a TEMPF format temperature file (*From File TEMPF*), or by using a TEMPF format solution result from a selected folder (*From Results Directory*). The TEMPF format in the simplest case is a list of element numbers with corresponding temperature values. If you use this command to specify temperatures in multi-layer shells in the *Multi-Layer Shells* group, you need to specify the layer to set up a boundary condition.

5.4. Solution examples

5.4.1. Infinite pipe heating problem

This problem considers the heating of a pipe that is clad in external thermal insulation. A temperature value is specified for the internal surface of the pipe and the external surface of the insulation layer. The pipe is infinitely long. The objective is to obtain a steady-state temperature distribution in the radial direction and to compare it with an existing analytical solution.

The problem can be solved in two ways: in classical two-dimensional formulation and in two-dimensional formulation with cyclical symmetry conditions.

The first stage of the solution involves constructing the geometry of the simulation region. To simplify creating an ordered FE mesh, in this problem the geometry model is divided into sectors (Figure 5.26, A), and appropriate splitting parameters are specified for these sectors (Figure 5.26, B). The FE mesh is constructed using the **2D Mapped Mesh** tool. Since a 2D mesh is used for the simulation, longitudinal linear dimensions of the geometry model are irrelevant.

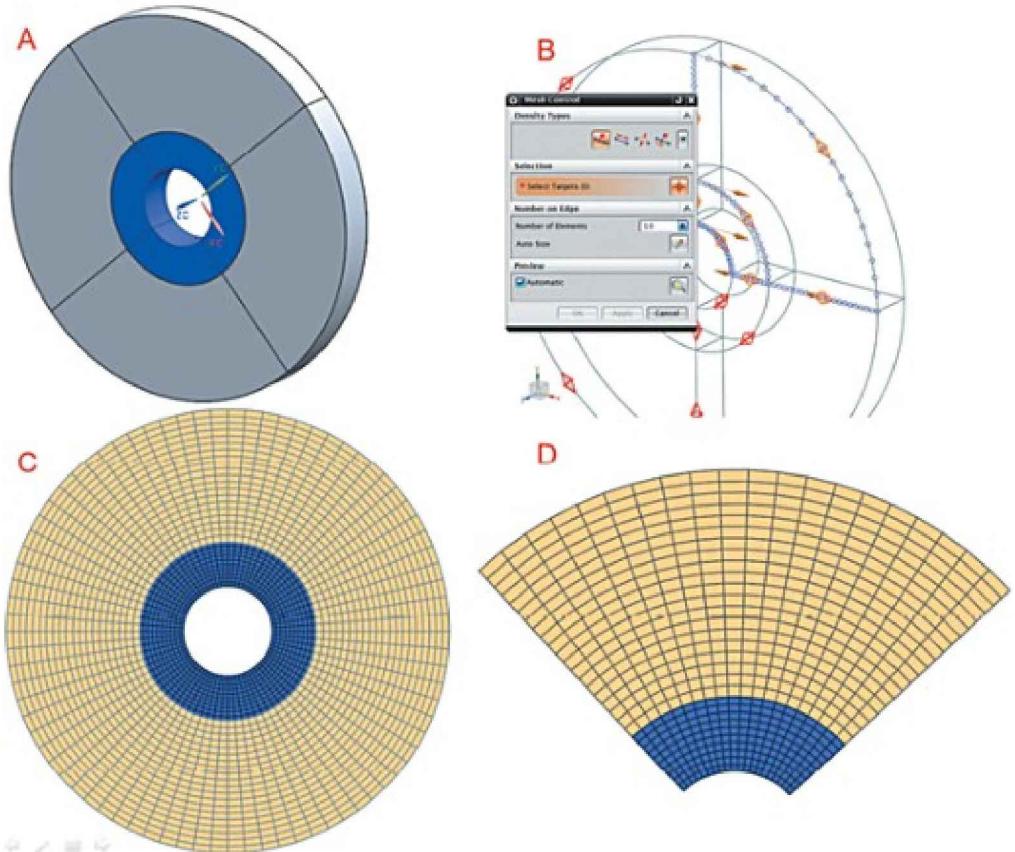


Figure 5.26. Geometry of the simulation region (A), splitting the region to construct a FE mesh (B), simulation mesh for full (C) and cyclically symmetric (D) formulations

It is assumed that the geometry of the simulation region and the FE model are already constructed. Figure 5.26 shows mesh models of the full (C) and symmetrical (D) formulations.

This problem is solved in steady-state formulation so the main parameter that influences the temperature distribution is the thermal conductivity of the material. The pipe is made of steel with a thermal conductivity coefficient $k=19 \text{ W/(m}\cdot\text{K)}$, and has asbestos insulation with thermal conductivity coefficient $k=0.2 \text{ W/(m}\cdot\text{K)}$. The internal section of the pipe has the radius $r = 0.01[\text{m}]$, while the external radius of the thermal insulation is $r = 0.05[\text{m}]$. The thickness of the pipe wall is $\Delta r = 0.01[\text{m}]$.

The following is the analytic solution of the problem. In accordance with equations for the cylindrical wall, the thermal resistance is of the form [12]:

$$R_{t,steel} = \frac{\ln(r_2/r_1)}{2\pi k_{steel}} = \frac{\ln(0.02/0.01)}{2\pi(19)} = 5.802 \times 10^{-3} \text{ (m}\cdot\text{K)/W}$$

$$R_{t,asb} = \frac{\ln(r_3/r_2)}{2\pi k_{asb}} = \frac{\ln(0.05/0.02)}{2\pi(0.2)} = 0.729 \text{ (m}\cdot\text{K)/W}$$

The heat flux is thus determined as follows:

$$\dot{q}_{tot} = \frac{T_{s,int} - T_{s,ext}}{\sum R_t} = \frac{600 - 100}{5.802 \times 10^{-3} + 0.729} = 680.30 \text{ W/m}$$

The temperature distribution along the steel portion of the pipe depending on the radial coordinate r (where $0.01 \leq r \leq 0.02$) is calculated as follows:

$$T(r) = T_{s,int} - \frac{\dot{q}_{tot} \ln(r/r_1)}{2\pi k_{steel}} = 600 - \frac{(680.30)\ln(r/0.01)}{2\pi(19)}$$

For the thermal insulation cladding (where $0.02 \leq r \leq 0.05$) the relation is of the form:

$$T(r) = T_{s,int} - \dot{q}_{tot} \left[\frac{\ln(r_2/r_1)}{2\pi k_{steel}} + \frac{\ln(r/r_1)}{2\pi k_{asb}} \right] = \\ = 600 - 680.30 \left(5.802 \times 10^{-3} + \frac{\ln(r/0.02)}{2\pi(0.2)} \right)$$

The resulting analytical expression defines the radial temperature distribution for both portions of the pipe.

The analytical solution of the problem holds true for a pipe of infinite length. In terms of NX Thermal this corresponds to setting adiabatic wall conditions at end faces of the simulation region (that is, the heat flux is distributed only in the radial direction). Since all boundaries of the simulation area where no boundary conditions are specified have the adiabatic wall condition applied automatically, you do not need to specify any boundary conditions. Specify the temperature of the internal and external boundaries of the simulation region (Figure 5.27, A). In this problem, the temperature of the external surface of the pipe is 600 °C, while the external surface temperature of the

insulation cladding is 100 °C.

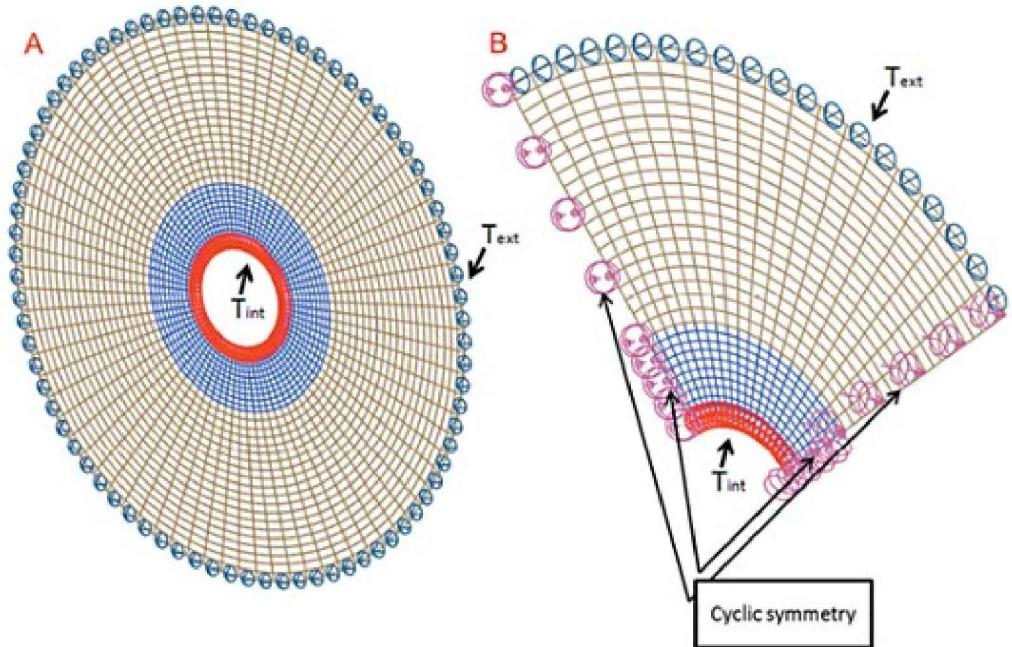


Figure 5.27. Boundary conditions for classical (A) and cyclically symmetric (B) formulations

For the variant with cyclical symmetry, you need to specify additional conditions (Figure 5.27, B). To do so, on the **Advanced Simulation** toolbar, select **Simulation Object Type→Thermal Coupling – Advanced→Perfect Contact**. In the **Parameters** group, select **Connection Method** of the **Primary – Revolved Secondary** type, specify a vector and a point of revolution (Figure 5.28).

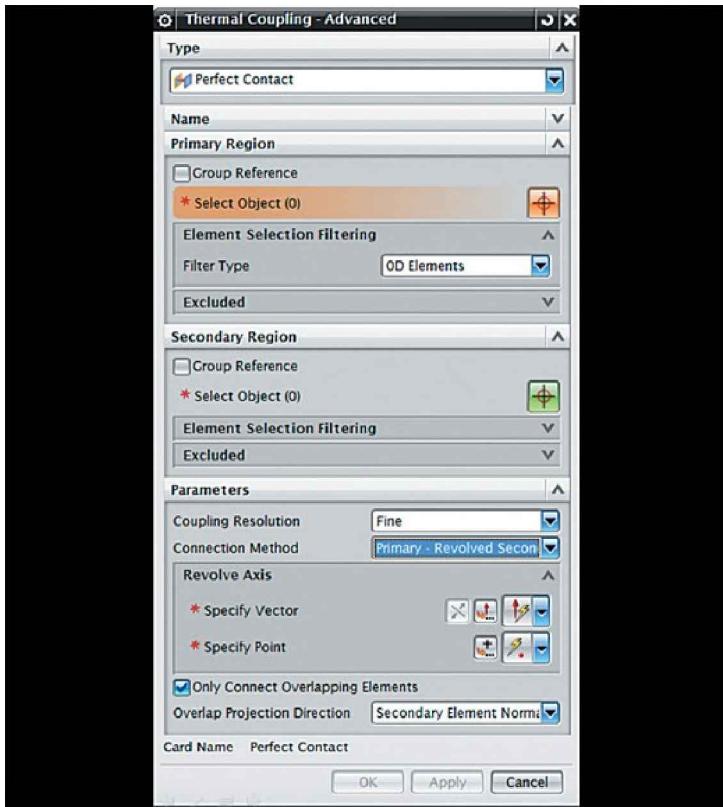


Figure 5.28. Specifying cyclical symmetry conditions

The main part of the formulation is now completed, and you can run the simulation of the problem. Simulation results can be represented in various forms, for example, as a temperature field (Figure 5.29, A) or a radial temperature distribution graph (Figure 5.29, B).

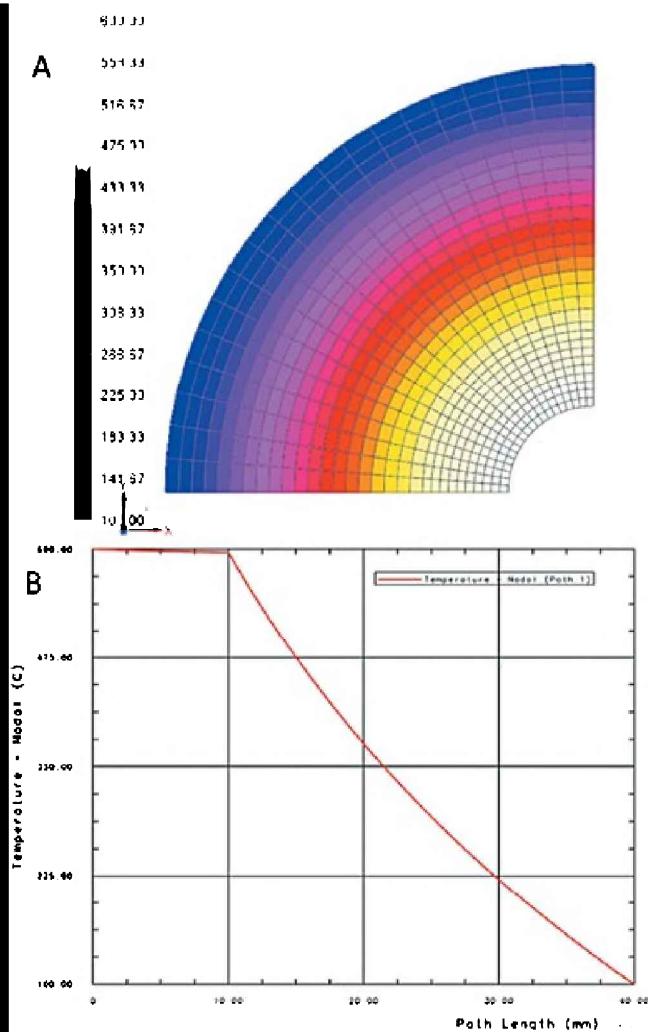


Figure 5.29. Simulation results represented as a field (A) and a radial temperature variation graph (B)

Numerical and analytic solution results are compared in the following table. The error of the numerical result does not exceed 0.24% in full as well as in cyclically symmetric formulation.

Table. Comparison of numerical and analytic solution results

r, m	$\text{ideal}, ^\circ\text{C}$	$T_{\text{full}}(r), ^\circ\text{C}$	$T_{\text{symm}}(r), ^\circ\text{C}$	$\Delta T_{\text{full}}, \%$	$\Delta T_{\text{symm}}, \%$
0,010	600,00	600,00	600,00	0,00	0,00
0,015	597,69	597,69	597,69	0,00	0,00
0,020	596,05	596,04	596,04	0,00	0,00
0,025	486,19	485,83	485,83	-0,07	0,07
0,031	367,60	367,25	367,25	-0,09	0,09
0,035	293,10	292,74	292,74	-0,12	0,12
0,040	227,62	227,26	227,26	-0,16	0,16
0,046	151,06	150,69	150,69	-0,24	0,24
0,050	100,00	100,00	100,00	0,00	0,00

5.4.2. Heat transfer by radiant heat exchange

This is a problem of heat exchange between two perpendicular plates joined along an edge (Figure 5.30, A). There is only radiant heat exchange (no heat conduction at the contact edge). The plates are immersed in infinite space with ambient temperature of 300 K. The emissivity factor of the first plate is $\epsilon = 0.6$, and its temperature is 1000 K. The emissivity factor of the other plate is $\epsilon = 1$. The temperature of the second plate is the unknown value. The second plate can exchange heat only from the side that faces the first plate (the other side is insulated). The first plate emits radiant energy towards the second plate. The plates are square in shape with a side of 0.5 m.

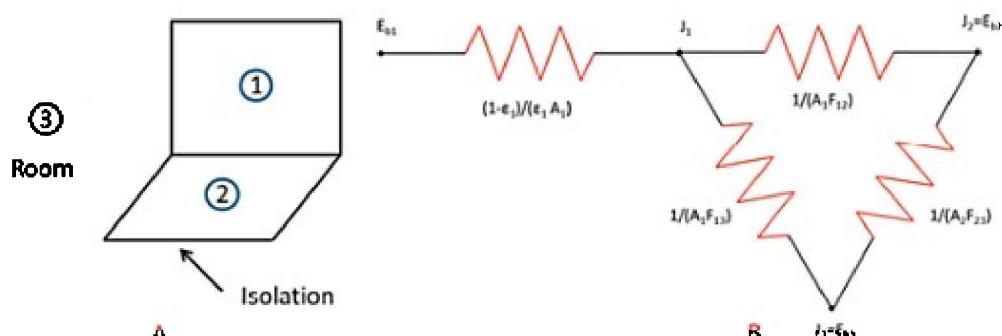


Figure 5.30. Simulation region (sketch) (A) and representation of the solution as “current in the circuit” using the Oppenheim method (B)

In accordance with the procedure proposed in [13], the problem in this formulation can be represented as a schematic (Figure 5.30, B), which describes “current” in a circuit with corresponding parameters [12]. Callouts: F_{ij} is the part of energy that reaches surface j after being emitted by surface i . A is the surface area [m]. J_i is the total radiation that is emitted by surface i in a unit of time per unit of area [W/m²], with reflected energy taken into account. E is the radiated power [W/m²].

In this formulation [12]:

$$F_{12} = 0.2 = F_{21}; F_{11} = F_{22} = 0; F_{12} + F_{13} = 1.0; \\ F_{13} = 1 - 0.2 = 0.8 = F_{23}$$

$$A_1 = A_2 = (0.5)^2 = 0.25$$

Resistance can be calculated as

$$\frac{1 - \varepsilon_1}{\varepsilon_1 A_1} = \frac{0.4}{(0.6)(0.25)} = 2.667$$

$$\frac{1}{A_1 F_{13}} = \frac{1}{A_2 F_{23}} = \frac{1}{(0.25)(0.8)} = 5$$

$$\frac{1}{A_1 F_{12}} = \frac{1}{(0.25)(0.2)} = 20.0$$

Also:

$$E_{b_1} = (5.669 \times 10^{-8})(1000)^4 = 5.669 \times 10^4$$

$$J_3 = E_{b_3} = (5.669 \times 10^{-8})(300)^4 = 459.2$$

The general schematic of sequential-parallel connection in accordance with Figure 5.30, B can be transcribed as

$$q = \frac{E_{b_1} - E_{b_3}}{R_{\text{equiv}}}$$

Then:

$$R_{\text{equiv}} = 2.667 + \frac{1}{\frac{1}{5} + \frac{1}{(20+5)}} = 6.833$$

$$q = \frac{56690 - 459.2}{6.833} = 8.229$$

Heat transfer is represented as

$$q = \frac{E_{b_1} - J_1}{(1 - \epsilon_1)/\epsilon_1 A_1}$$

Therefore, $J_1 = 34,745 \text{ W/m}^2$.

The J_2 value is defined through the relation of resistances J_1 and J_3 :

$$\frac{J_1 - J_2}{20} = \frac{J_1 - J_3}{20 + 5}$$

and

$$J_2 = 7613 = E_{b_2} = \sigma T_2^4$$

The final temperature relation is:

$$T_2 = \left(\frac{7316}{5.669 \times 10^{-8}} \right)^{1/4} = 599.366 [K]$$

To solve this problem using NX Thermal, you need to construct a simulation region that correspond to the sketch (Figure 5.30, A), and the simulation mesh. Each of the perpendicular planes is described by a single mesh 2D element. Since only radiant heat exchange mechanisms are relevant in this

problem, and physical properties of materials are not important, the elements are defined as zero-thickness shells with defined thermo-optical properties (Figure 5.31, A). Thermo-optical properties are applied only to those sides of elements that participate in radiant heat exchange.

When you specify boundary conditions, you need to set the temperature of the first wall to a fixed value of 1000 K and select the **Radiation** simulation object type. Specify *Element Subdivision 5 (Very Precise)*. Specify *Calculation Method – Hemicube Rendering* in accordance with parameters shown in Figure 5.31, B.

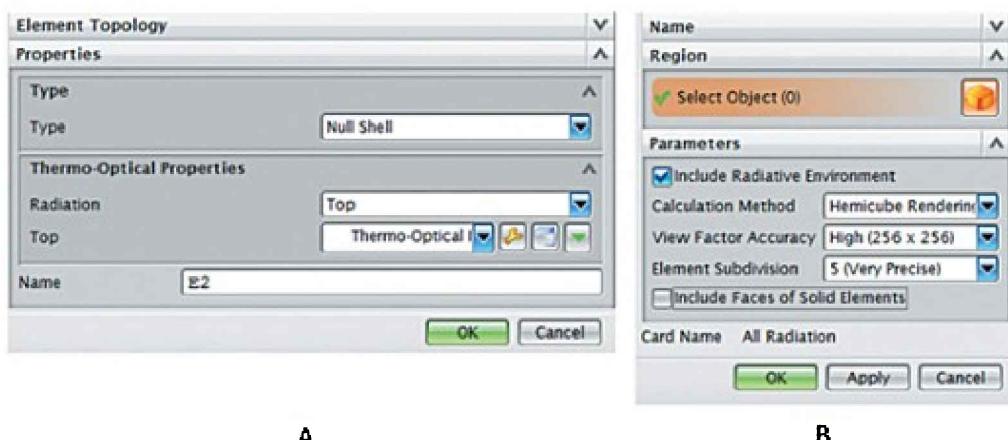


Figure 5.31. Specifying the collector properties (A) and setting up the radiant heat exchange simulation algorithm (B)

Run the solution of the problem. As a result of the simulation the temperature value T_2 is found to be 599.330 K, therefore, the relative error with regard to analytic solution is under 0.025%.

5.5. Accounting for mass transfer in thermal problems

In some thermal problems, the flow rate of the heat-transfer fluid must be accounted for. It is difficult to accomplish with standard approaches to solving heat exchange problems. To make this easier, NX Thermal introduces the *Duct 1D* element, which is in fact a solution of one-dimensional fluid dynamics equations. This element accounts for the variation of flow parameters along the specified path but it does not take into consideration any variation of parameters in element cross-section.

Each *Duct* type element is a combination of two sets of equation. The first set defines the thermal state and heat exchange with the environment. The second set defines fluid dynamics parameters: flow rate, pressure, and hydraulic resistance values. The thermal and fluid dynamics solutions are carried out simultaneously. Elements are divided into two components at the level of the solver, and you do not have to do anything.

To construct a 1D duct you only need a spatial or planar curve that represents the path of the fluid along the duct. If the shape of the duct is important (for example, if it determines the flow rate), you need to select or create a cross section profile. In addition, you need to specify the fluid that flows in this duct. The duct itself is specified with the **1D Mesh** command of the **Advanced Simulation** toolbar.

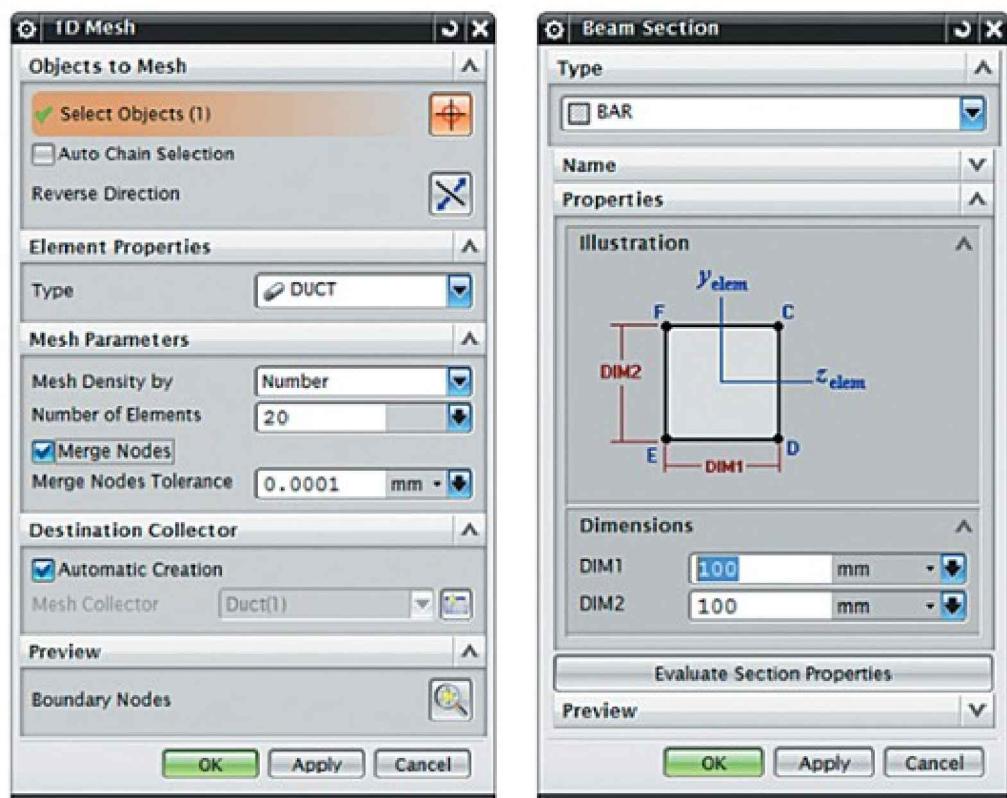


Figure 5.32. Constructing a 1D mesh and specifying duct profile

When you create the 1D mesh collector (Figure 5.32) that is used as the *Duct* type object, select one of the available types: *Duct*, *Duct with Thick Wall*, *Duct with Mass Flow*, *Duct Axisymmetric*, *Duct with Mass Flow*

Axisymmetric.

Specifying the properties of the fluid involves selecting the material of the medium flowing along the duct *Fluid Material* from the materials library.

- *Duct* is a duct with a defined cross section profile regardless of the material of the profile (you specify properties of the fluid, and the shape and size of the flow opening).
- *Duct with Thick Wall* is a duct with the specified geometry of the flow opening profile, which has walls of defined thickness. The material of the duct is also specified.
- *Duct with Mass Flow* specifies a duct with a defined property of the flowing fluid (only properties of the flowing fluid are specified). The channel does not have a defined profile shape.
- *Duct Axisymmetric* is a duct with a specified property of the flowing fluid. The shape of the duct can be described as axisymmetric. You need to specify the radius of the open section, the coordinate of the duct centre, and the axis of revolution.
- *Duct with Mass Flow Axisymmetric* allows modelling a duct with a defined property of the flowing fluid whose shape can be described as axisymmetric. You need to specify the coordinate of the duct center, and the axis of revolution.

After creating the 1D ducts, you can specify boundary conditions for these ducts. You can set boundary conditions using the ***Duct Flow Boundary Conditions*** command.

Flow in the duct is most easily specified using the *Duct Fan/Pump tool*. With this tool, you can specify the flow *Velocity*, *Mass Flow*, or *Volume Flow Rate*, as well as *Pressure Rise*. You can use the *Pressure Rise* condition to model a device that causes the fluid to flow (such as a pump or a fan). To do so, specify the head and rate relation as the initial data. The head and rate data can be described by a formula or a table.

5.6.1. Specifying the head and flow rate relation

The following is a description of how you can specify head and rate in table form. To do so, as the *Mode* select *Pressure Rise*, and in the *Pressure Rise* list, select *Field*. Then in the *Specify Field* list, select *Table Constructor...* as

shown in Figure 5.33.

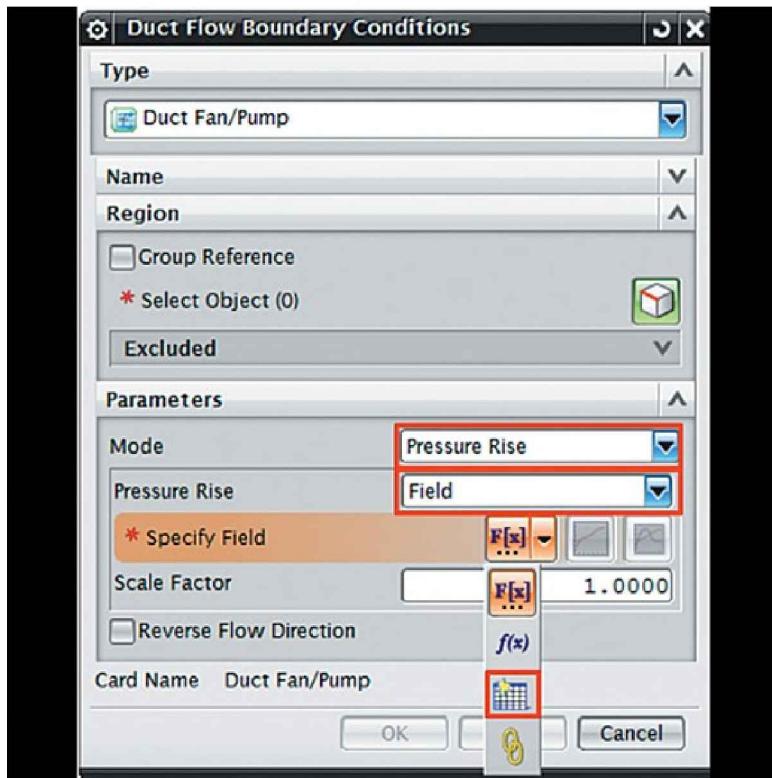


Figure 5.33. Specifying the head and flow rate relation

In the *Independent* tab of the table constructor, in the *Domain* list, select *1-D General* and specify, which of the independent variables is to be used: *Mass Flow Rate* or *Volume Flow Rate*. If necessary, in the *Variables* group, you can change *Units* to the required units of measurement (Figure 5.34).

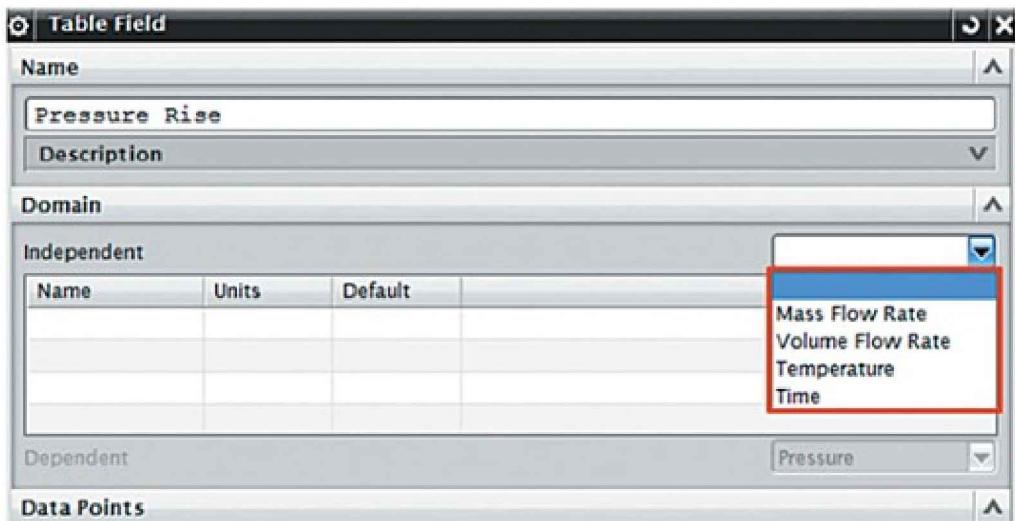


Figure 5.34. Selecting independent variables to specify the head and flow rate relation

You can populate the initial data table in the *Table Field* dialog box or use an Excel spreadsheet. In the latter case, click the *Edit table in spreadsheet* button. The spreadsheet application will launch, where you can enter the initial data in the relevant columns. An important operation is importing the Excel data into NX. To do this, click the *Reload Table* in the *Add-Ins* tab of the Excel spreadsheet. After the reload, the corresponding columns in the *Table Field* menu are populated (Figure 5.35).

If you populate the table in the *Table Field* dialog box, the data are entered in the initial data entry box (in the very bottom of the dialog box). To confirm data entry, click the *Accept Edit* button with the green check mark to the right of the data entry box. You need to specify two numerical parameters: flow rate and pressure.

To visualize the created relation, use the *Plot (XY)* tool after returning to the **Duct Flow Boundary Conditions** dialog box.

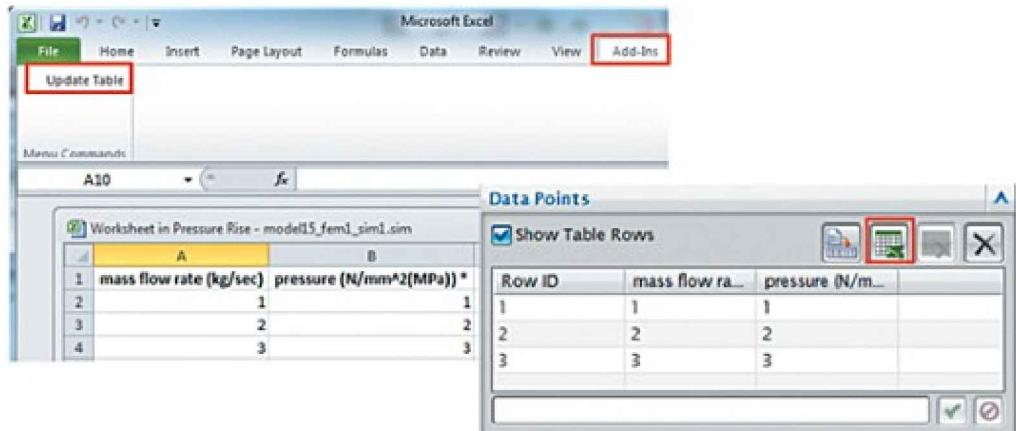


Figure 5.35. Creating a head and flow rate relation in an Excel spreadsheet

Head losses can be specified for any elements of the duct (not only at the outlet). To set up losses, use the *Duct Flow Properties* type and specify the point on the sketch (or FE mesh) of the channel where a head loss is specified.

To create local head loss using the *Head Loss* tool, do the following:

1. Select the boundary condition type in the *Duct Flow Boundary Conditions* dialog box (in the *Duct Flow Properties* group). In the *Wall Friction* list, select *Head Loss Override*, as shown in Figure 5.36. Under *Head Loss*, click *Create Modeling Object*.
2. Select one of the methods of specifying losses: *Specify*, *Square-Edge Inlet*, *Well-Rounded Inlet*, *Protruding Inlet*, *Outlet*, *Contraction*, *Expansion* (Figure 5.36). In the latter two cases, the system proposes a template that calculates losses from known parameters of the duct.
3. Select the object for which the head loss coefficient is to be specified.

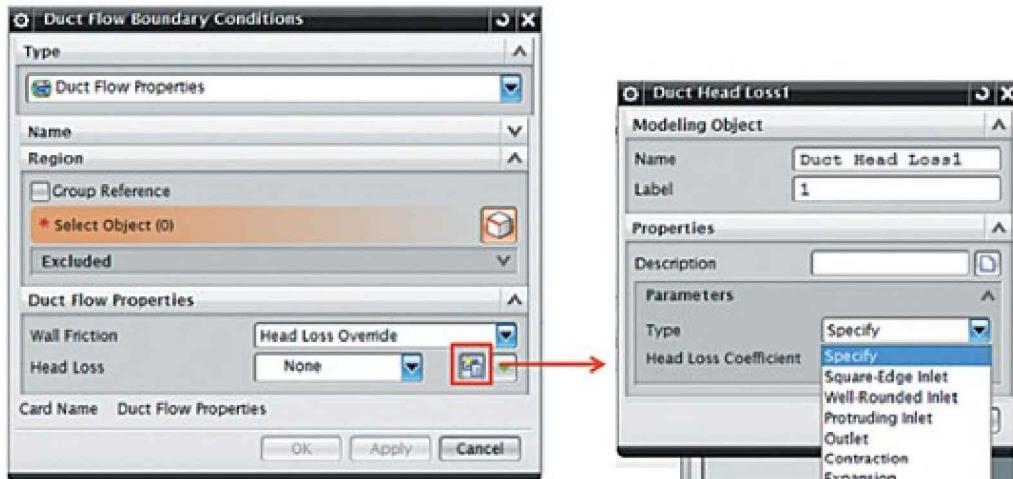


Figure 5.36. Specifying local head losses

5.6.2. Solving the problem of heat-transfer fluid flow in a heat exchanger tube

This problem analyzes the flow of heat-transfer fluid in a heat exchanger tube. The tube has a finned surface. The cooling is achieved by free convection from the outer surface of the heat exchanger. The tube is made of steel, while the fins are made of copper. The heat-transfer fluid is water, its inlet temperature is 95 °C. The ambient temperature is 20 °C.

A segment of the heat exchanger is considered. This segment contains three fins. A sketch of the model and preliminary subdivision of the body (for building a structured FE mesh) are shown in Figure 5.37.

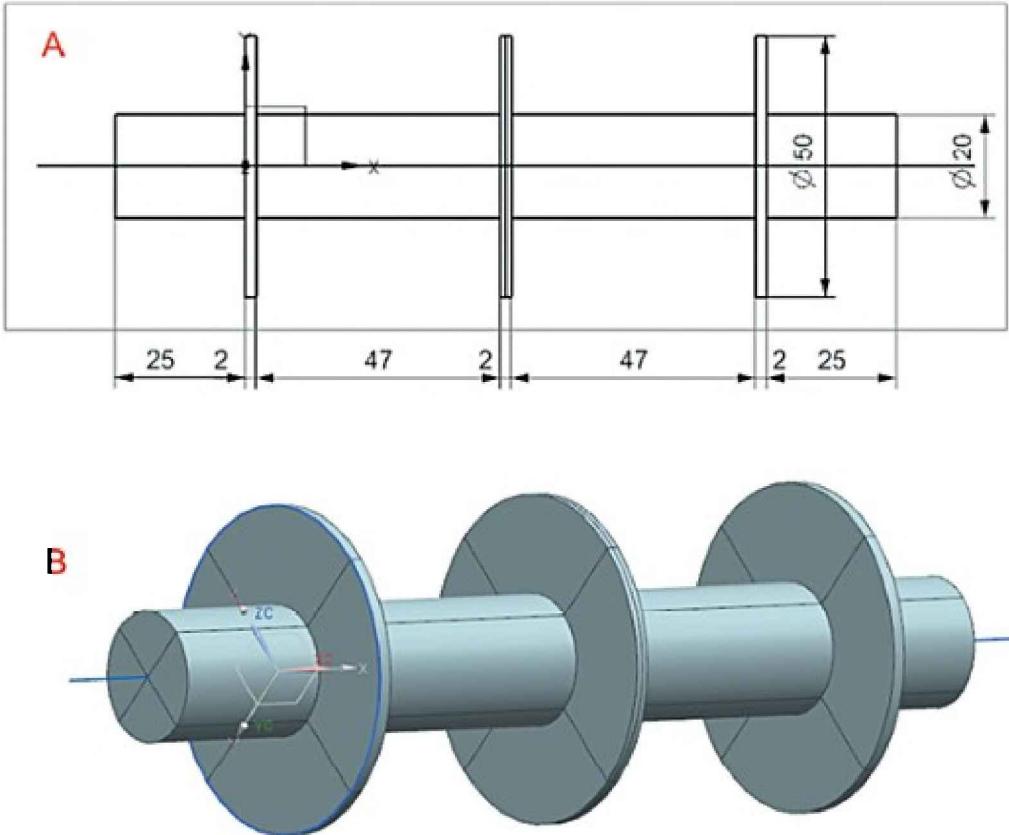


Figure 5.37. Sketch of the model (A) and subdivision of the body (B) for building the FE mesh

A 1D duct element is used to account for flow of the heat-transfer fluid. Metal elements are described using a 2D mesh. Thicknesses are accounted for using properties of 2D elements. As a result of this representation, the FE model contains only 1D and 2D elements.

Now that the geometry is prepared, you can construct the FE mesh. First set up the mating of geometry bodies that were created during model decomposition. This is a prerequisite of constructing the FE model without gaps at joints of decomposed parts. To set up mating, use the **Mesh Mating Conditions** tool. Use *Automatic Creation* and select the entire model as the mating objects. Use *Glue-Coincident* mating type (*Figure 5.38*).

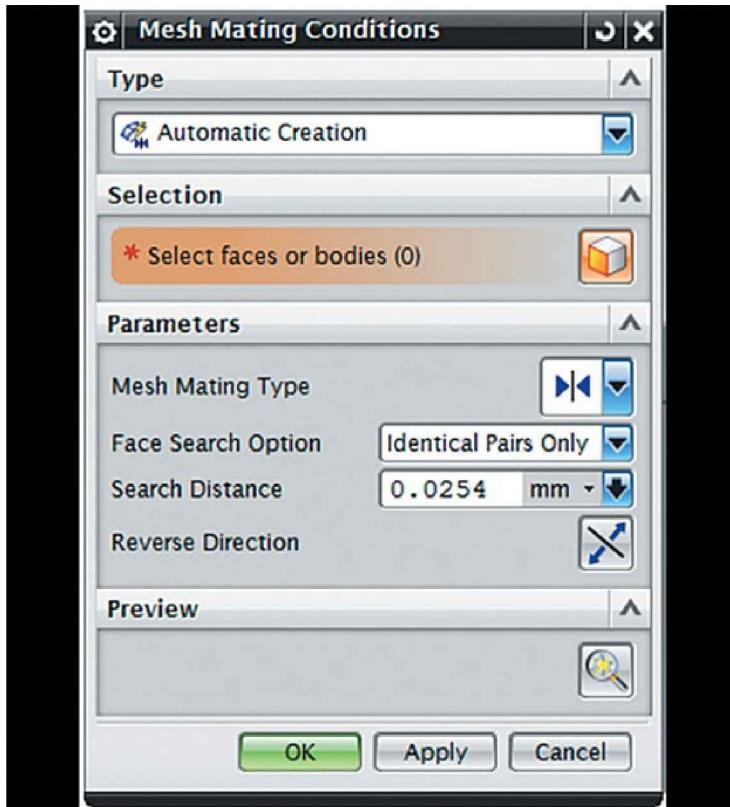


Figure 5.38. Body mating parameters for FE mesh mating

Now define splitting parameters on model edges to subsequently create a regular mesh based on these edges. To do so, use the **Mesh Controls** dialog box and the *Number on Edge* group (Figure 5.39). The splitting of internal and external circular edges must match.

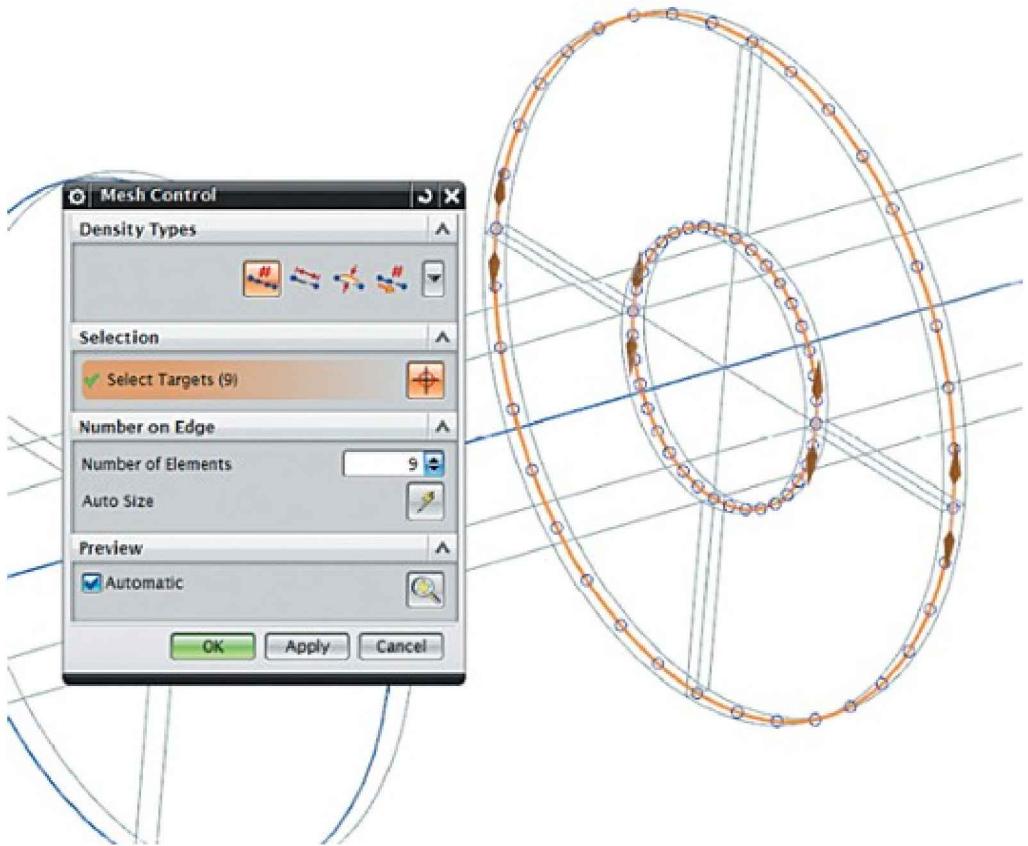


Figure 5.39. Specifying edge splitting

After specifying splitting of all edges in this way, construct the regular mesh. To do so, use the *2D Mapped Mesh* command. In the dialog box (Figure 5.40), select the *QUAD4 Thin Shell* element type, the collector, and objects for which you need to build the mesh. Select the *Export Mesh to Solver* check box. Since this model uses two materials, create two corresponding FE mesh collectors.

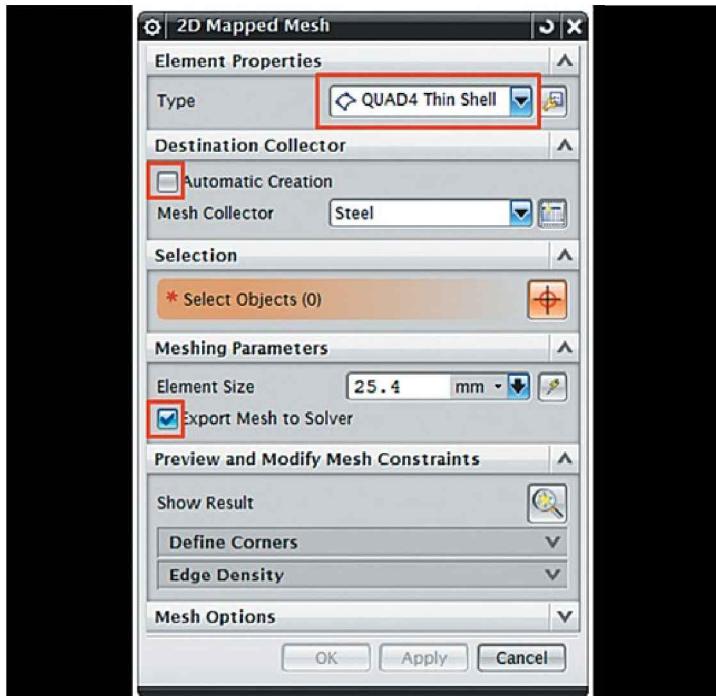


Figure 5.40. Creating a 2D regular mesh

An example of creating copper and Steel collectors is shown in Figure 5.41.

Specify the necessary thicknesses. In this example the thickness of copper fins is 1 mm, and the steel wall thickness is 3 mm.



Figure 5.41. Creating mesh collectors

After creating the collectors, click **OK** to return to the **2D Mapped Mesh** dialog box, and select surfaces for which you want to build the mesh with material properties of the corresponding collector. Specify the size of finite

elements, and click **OK** to complete the command. The FE mesh is constructed.

The stages described above are performed for all geometry elements that require numerical modeling. As a result, you should end up with two 2D mesh collectors. The number of meshes in every collector can differ depending on the method of construction but must not be less than one mesh per collector (Figure 5.42).

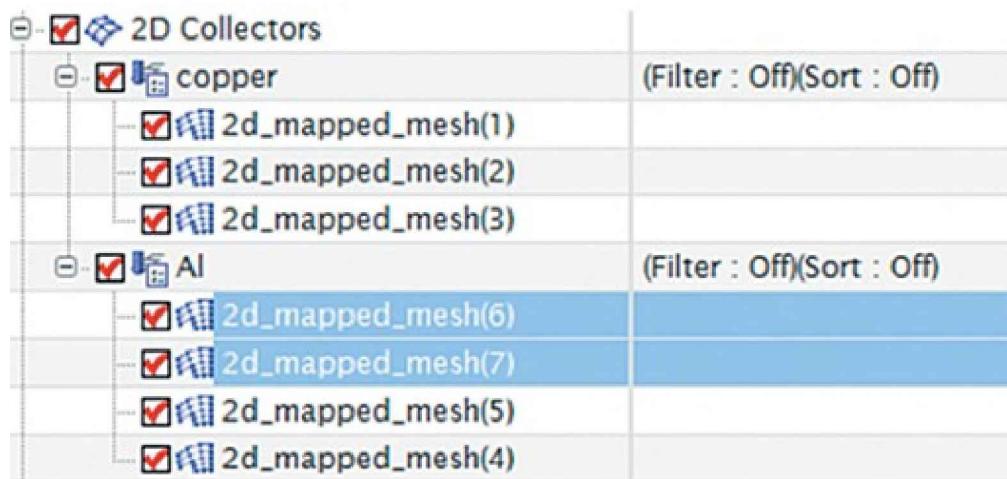


Figure 5.42. Created FE meshes in collectors

Now construct the 1D mesh that describes the flow of heat-transfer fluid. You use the same options as for the 2D mesh: specify geometry for which the mesh is being constructed, select the type of the 1D mesh, specify splitting parameters, and create the collector that describes physical properties.

The sketch curve that coincides geometrically with the revolution axis of the model is used as the geometry for which you build the 1D mesh. Since you need to model flow, select *Duct* as the 1D element type. Specify 50 lengthwise subdivisions for this problem. Select the flow material (Water) from the materials library, a constant cross section type, a circular profile, and a radius of 10 mm (Figure 5.43).

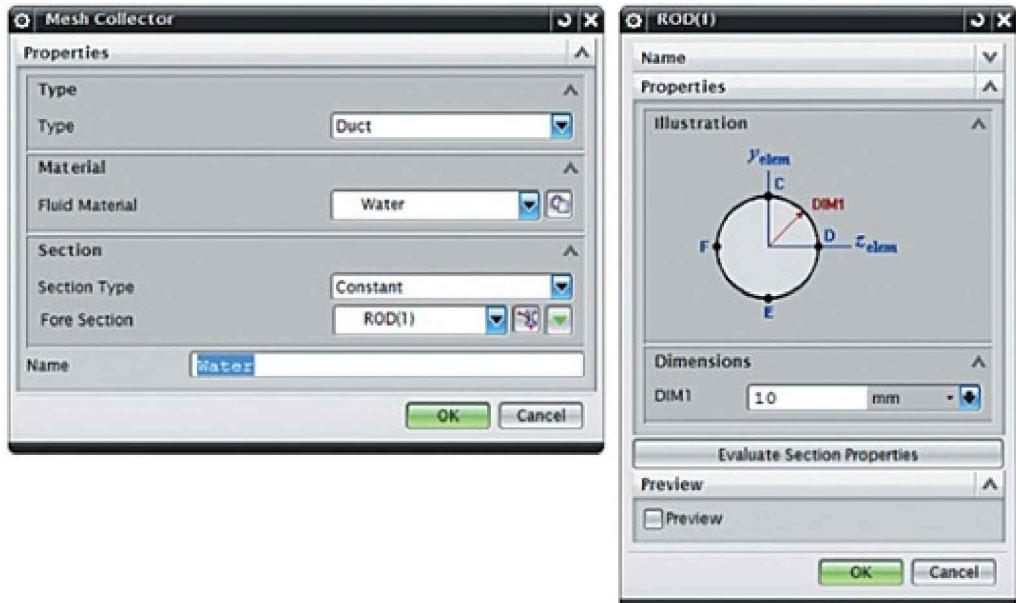


Figure 5.43. Creating a collector that describes the duct and the heat-transfer fluid

If necessary, you can display the created model in 3D representation. Figure 5.44 shows a visualization example.

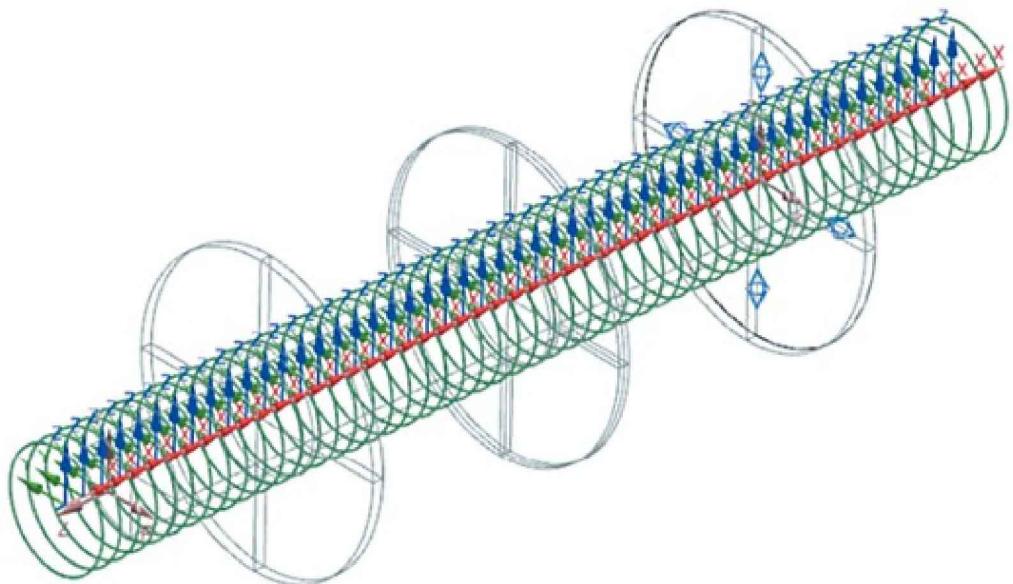


Figure 5.44. Visualization of 1D elements

The result of operations in the FEM is a FE model similar to the one shown in Figure 5.45.

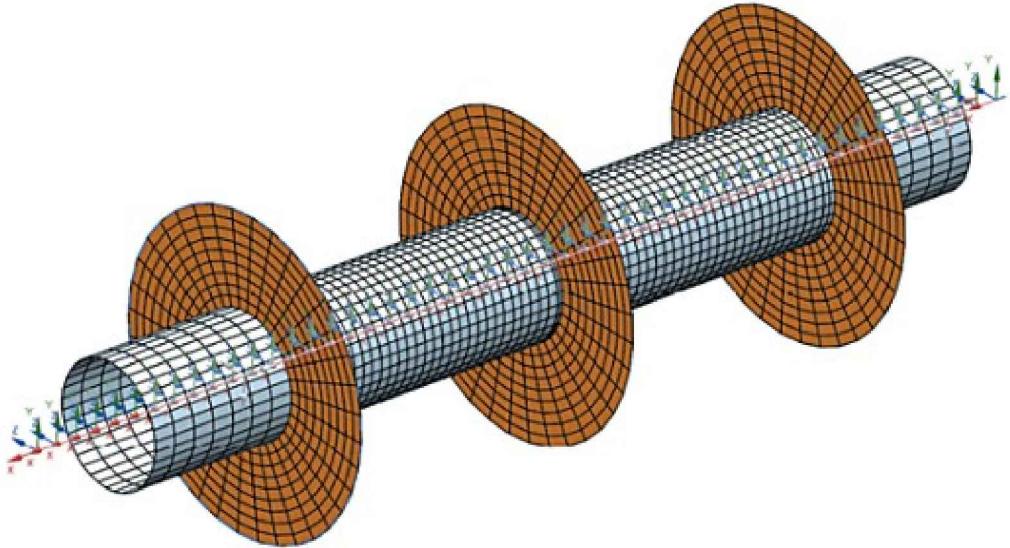


Figure 5.45. The developed FE mesh

After creating the FE mesh, proceed to specifying boundary conditions in the S/M file. Modelling is carried out with the following assumptions: the flow of heat-transfer fluid in the duct is fully developed (fully developed boundary layer), heat exchange between the fluid and the inner wall of the duct occurs only due to forced convection, heat emission into the environment happens due to free convective flow, and radiant heat exchange is neglected.

Before you run the simulation of the problem, do the following:

1. Specify flow parameters in the duct. To do so, use the **Duct Flow Boundary Conditions** tool. Describe the flow using the velocity of the heat-transfer fluid, which is equal to $V = 0.01$ m/s. If necessary, reverse the flow direction by selecting the corresponding check box. Apply this boundary condition to the first 1D element (for convenience, use the element selection filter), as shown in Figure 5.46.

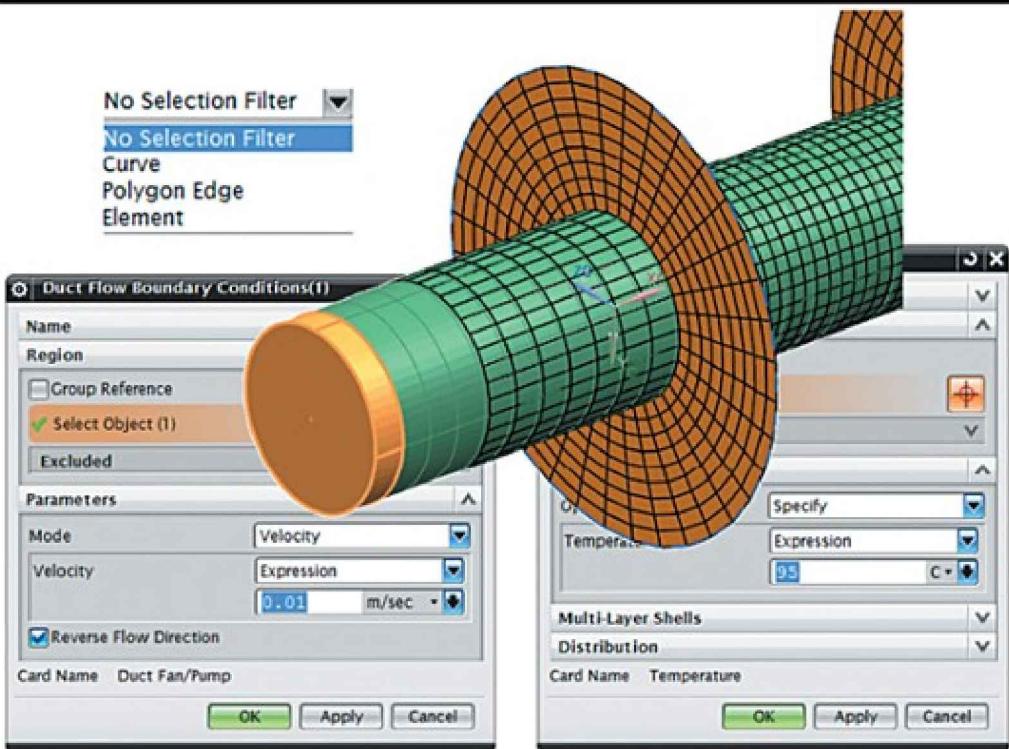


Figure 5.46. Specifying boundary conditions of the 1D duct

To specify temperature, use the **Temperature** tool. Specify 95 °C as the temperature value in the same channel element where you specified the velocity of the fluid (Figure 5.46). No conditions are applied at the outlet of the duct. This means that no losses are considered for the fluid exiting the region in question (an outflow condition with zero loss coefficient is automatically defined).

2. Thermal coupling between the heat-transfer fluid in the duct and duct walls is implemented as forced convection in accordance with the formulation of the problem. Use the command: **Simulation Object Type→Simulation Object Type Thermal Coupling – Convection**. Specify convection Type: *Forced Convection Coupling*. To create the coupling, in the *Convecting Region* list select duct walls, and in the *Fluid Ducts* list, select the line running along the axis of the duct. In accordance with the assumptions of the *Parameters* group, select the *Duct Fully Developed Flow* flow type. Select the *Only Connect Overlapping Elements* check box. Click **OK**.
3. Heat is removed from the finned pipe only by free convection. For some

geometry objects in NX Thermal, there are heat exchange correlations based on empirical relations. For this problem, cylinder and inclined plate correlations are used. The former correlation describes convection at the outer surface of the pipe, and the latter correlation describes convection at the surface of fins. The diameter of the pipe is specified with wall thickness (26 mm) included, and both sides of fins are accounted for. Ambient parameters: air under normal conditions *Ambient Condition*. To specify the boundary condition, use the **Convection to Environment** command as shown in Figure 5.47.

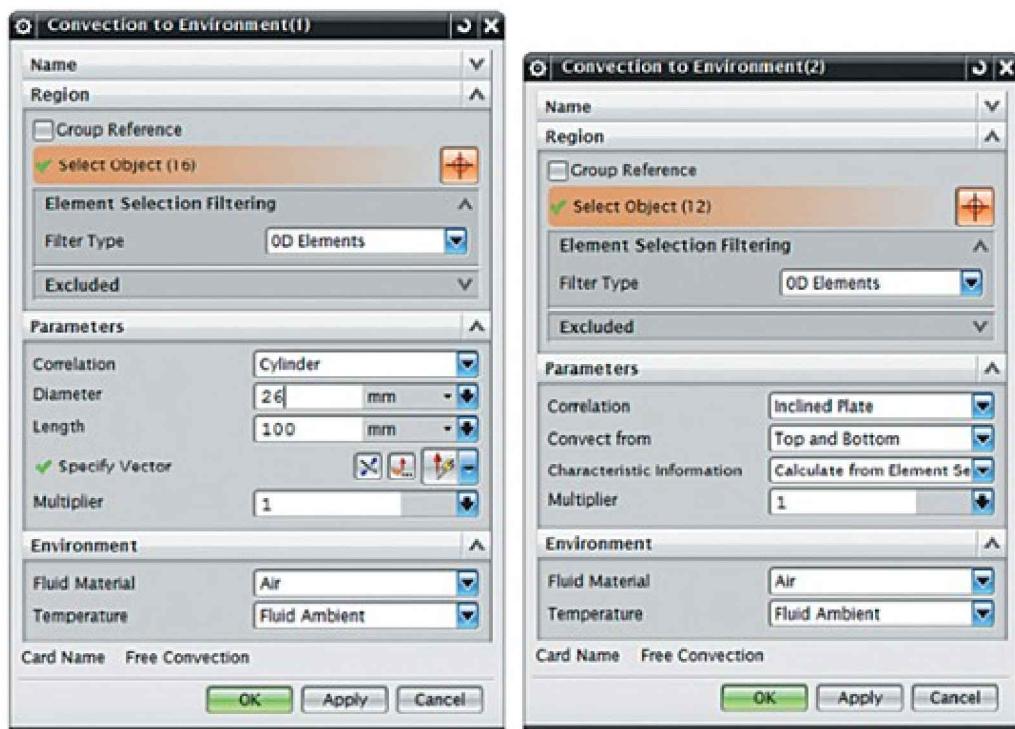


Figure 5.47. Setting up convection to environment

Run the simulation.

When the simulation finishes to analyse results, right-click *Results* in the model tree to switch to the post processor with the results loaded. To view the simulation results, double-click the solution tree to select the relevant node, for example *Temperature – Elemental*.

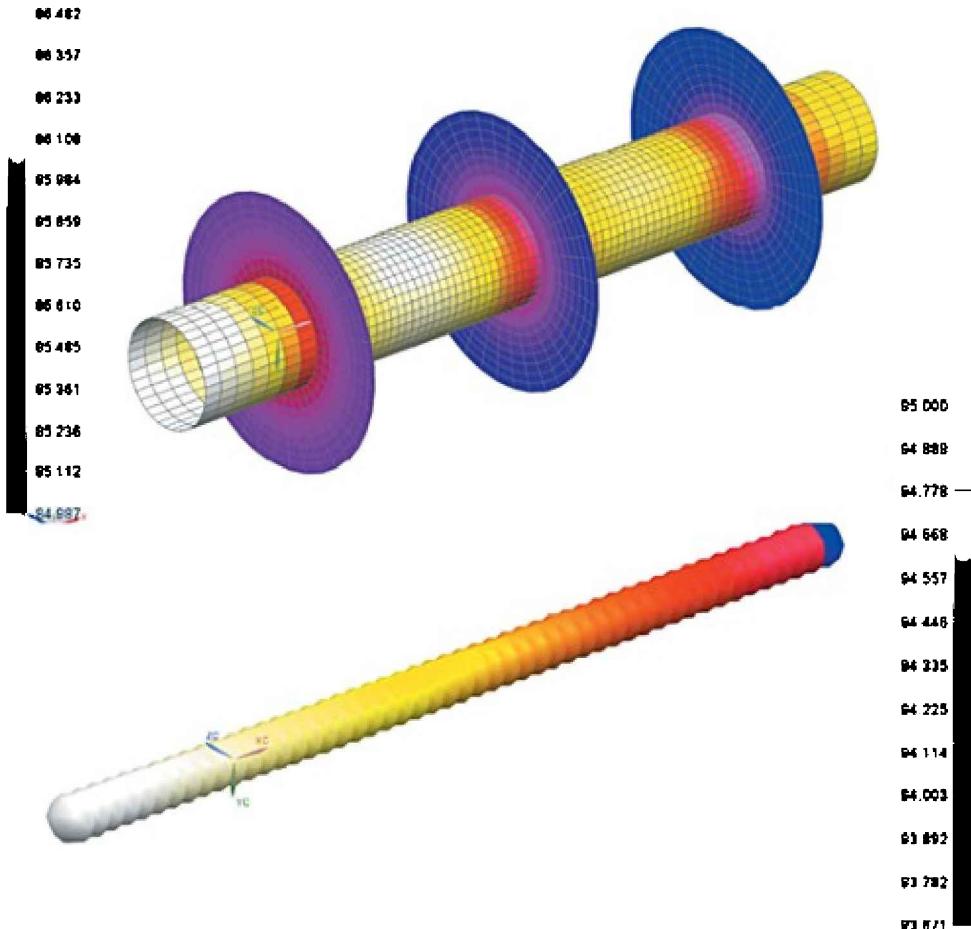


Figure 5.48. Temperature of the solid-body part of the model and the heat-transfer fluid

Since the model includes both 1D and 2D elements, some results, such as temperature, can be analysed separately for each type. To select the type of elements to display in the *Fringe Plots* tree of the post processor, expand the *Post View* node and select or clear check boxes of the corresponding element types. Figure 5.48 shows the temperature field distribution in the solid-body part of the model and in the heat-transfer fluid.

Chapter 6. Modelling fluid dynamics processes

Computational fluid dynamics is one of the most important and computationally intensive branches of numerical analysis. Its applicability is extremely broad to various operation aspects of structures and devices.

The NX Flow module, as well as its advanced version, NX Advanced Flow, can solve a broad range of fluid dynamics problems. The solver is based on the Navier-Stokes equations. This approach represents the current state of the art in this branch of numerical analysis.

6.1. General capabilities of NX Flow and NX Advanced Flow

NX Flow is unique because of its tight integration with the NX environment. Integration with NX means design geometry passes seamlessly to the simulation environment, while giving analysts powerful geometry tools that significantly shorten CFD model preparation. Additionally, integration in NX streamlines multiphysics processes since the NX Flow modules are available in the same environment used for thermal, structural, and motion simulation. CFD is often used in conjunction with thermal analysis to evaluate performance of thermal management systems, such as used in electronic devices, and so the NX Flow modules can seamlessly integrate with the NX Thermal products to efficiently solve thermal management problems.

Basic capabilities of NX Flow include:

- Finite-volume CFD solver that uses an FE mesh.
- Analysis of steady-state and transient processes.
- A broad selection of turbulence models (fixed viscosity model, mixing-length model, $k-\epsilon$, $k-\Omega$, SST).
- Analysis of internal and external flows.
- Automatically join dissimilar fluid meshes at interfaces between different parts of complex assemblies. .
- Automatic generation of CFD meshes with local biasing and generation of boundary-layer mesh.
- Inlet/outlet boundary conditions, pumps, fans, screens, guide vanes, deflectors, and porous membranes.

- Accounting for linear and nonlinear boundary conditions.

Additional features of NX Advanced Flow include:

- Analysis of high-speed compressible fluid flow.
- Accounting for non-Newtonian fluid flow.
- Rotating coordinate systems.
- Accounting for motion of solid bodies.
- Active fan controllers.
- Particle paths tracking, scalars, particle transfer.
- Ability to use cyclical symmetry and periodicity conditions.
- Porous membranes.
- Hydraulic meshes (including 1D–3D connections).
- Parallel computing.
- Analysis of multi-component mixture flow.
- Analysis of multiphase flow.
- Accounting for humidity and condensation.
- Comfort assessment in accordance with ASHRAE standards.

This chapter discusses the features of NX Advanced Flow, therefore some of the solutions described here may be unavailable if you use NX Flow.

6.2. Constructing a fluid domain using surface wrap technology

Solving computational fluid dynamics problems includes challenges that are not present in structural and thermal problems. One challenge is the need for substantial modification of the geometry model to create a fluid domain. Since the design model describes the solid-body part of the structure, in the best case it delineates only the boundaries of the fluid domain. To construct a mesh using the “classical” method (Chapter 3, Part 1), you need solid-body geometry of not only the walls but also the whole fluid domain. This domain is not represented in the design model. Therefore, you need to create the interior part of the geometry or build the exterior fluid domain around the model immersed in the flow.

This problem is typically solved using Boolean operations. For example, you can create the geometry of the interior part of the model by subtracting the existing geometry from auxiliary geometry. For external flow problems, the

approach is analogous but the auxiliary geometry is constructed with consideration of the size of the surrounding domain. Figure 6.1 shows the most basic examples of applying this approach. In one case, an existing original design geometry (Figure 6.1, A) is used to construct a domain for the internal flow problem (Figure 6.1, B), and in the another case, the same geometry is used for the external flow problem (Figure 6.1, C).

In some cases, Boolean operations are difficult to apply, and many additional steps are involved due to the particular geometry of the design model.

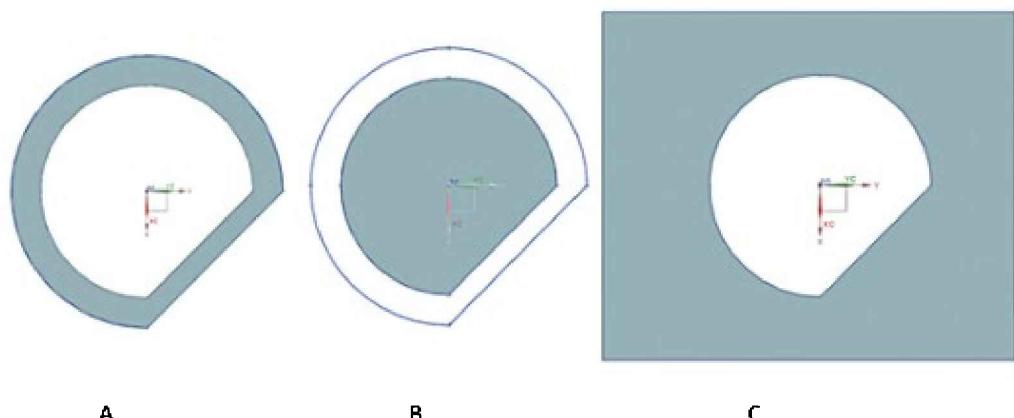


Figure 6.1. Using Boolean operations to create a fluid domain: A. Original design geometry. B. Fluid domain for the internal flow problem. C. Fluid domain for external flow

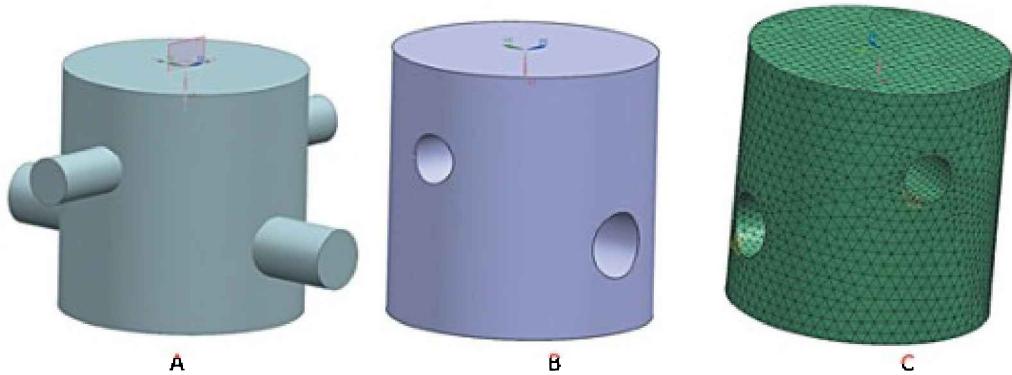
Moreover, you can use a specialized method to build the fluid domain geometry. This method is based on the technique that involves filling a volume with small auxiliary bodies. In particular, this method can be described as filling a volume with spheres of a given diameter. It is important that the body have closed boundaries (otherwise, the spheres would fill an infinite volume), and a starting point must be selected within the volume of the future fluid domain. This method is implemented by the **Create Surface Wrap Fluid Domain** tool. The algorithm this tool uses can be likened to pouring small spheres of identical diameter into a cavity. The smaller the sphere, the finer the areas it can get into. The sought body is conceived of as an extrapolation of the entirety of all spheres.

Figure 6.2 shows an example of building a simulation region using the **Surface Wrap Fluid Domain** technique. The problem involves a cylindrical

vessel that is penetrated by two cylindrical bodies (tubes). A fluid domain must be constructed that includes the vessel but excludes the tubes (a cross flow problem).

To do so, use the **Create Surface Wrap Fluid Domain** command. This command is available, for example, in the main menu: **Insert→Fluid Domain**. This command is available only for FE meshes of Thermal/Flow module models.

If you use the *Surface Wrap* method, select all bodies that need to be considered in building the simulation domain, and specify the point in space where the filling starts. As a result, you obtain the sought domain, depending on the size of the object that fills the model.



*Figure 6.2. Constructing a fluid domain using the Surface Wrap technique:
A. Original geometry. B. Result of domain construction. C. Simulation mesh
constructed for the resultant geometry*

Geometry created in this way (Figure 6.2, B) is used in constructing the FE mesh like any other polygon geometry (Figure 6.2, C).

Geometry constructed using the *Surface Wrap Fluid Domain* technique is not intended for subsequent editing.

6.3. Using the Fluid Domain technique to build meshes

When modelling problems with the NX Flow module, you can use both the standard FE mesh generator (described in Chapter 4 of Part 1) and the specially developed automatic fluid domain mesh generator. The latter generator was developed for solving fluid dynamics problems and is well-

adapted for this type of problems. It is available only in the NX Flow module.

The *Fluid Domain* meshing command in NX Flow was developed to automate processing of mesh models. The main advantage of this model generation method is automatic localization of the fluid domain. If the simulation domain includes objects that are impermeable to the flow, such objects are automatically found and accounted for when the mesh is built. Another important advantage of this method is the ability to create prismatic mesh layers with defined parameters near selected objects. You can use this option to build a boundary layer mesh near walls, and it is constructed automatically. The wall boundary layer mesh can be based on a template 2D mesh built in the *FEM* file.

An example of using this technique is shown in Figure 6.3. To build the simulation mesh, on the **Advanced Simulation** toolbar, in the **Simulation Object Type** list choose the *Fluid Domain* command. In the *Fluid Domain Region*, select the original geometry as the object (Figure 6.3, A).

You can control the discretization of the model with the *Element Size* parameter.

One of the special aspects of the *Fluid Domain* parameters is the impossibility of viewing the simulation mesh before the problem is simulated, for example, with the *Write Solver Input File* option.

Therefore, to visualize the resultant mesh, you need to:

- Choose the *Solve* command in the shortcut menu to run the simulation.
- Select the *Write Solver Input File* option in the dialog box where you run the simulation.

When the operation is executed, the generation of the simulation mesh starts, after which a results file is created, containing only FE mesh data. It is shown in Figure 6.3, B.



*Figure 6.3. A. Selecting a body for building the mesh using the **Fluid Domain** tool. B. Resulting mesh*

To build a mesh that is adapted to flow near the wall, select the following parameters when creating it in the **Fluid Domain** dialog box:

- As the *Type* of the mesh, select *Fluid Surface Mesh*.
- In the *Region* group, specify the surfaces near which you need to generate adapted FE meshes.
- Select the *Create Boundary Layer Mesh* check box to make available the menu group that allows you to set up parameters of the wall-boundary mesh.
- Select the *Specify Local Surface Mesh Density* check box to override the parameters of the wall-boundary mesh that are subsequently used in generating the prismatic wall-boundary mesh. For example, you can construct differently biased meshes for different surfaces (Figure 6.4).

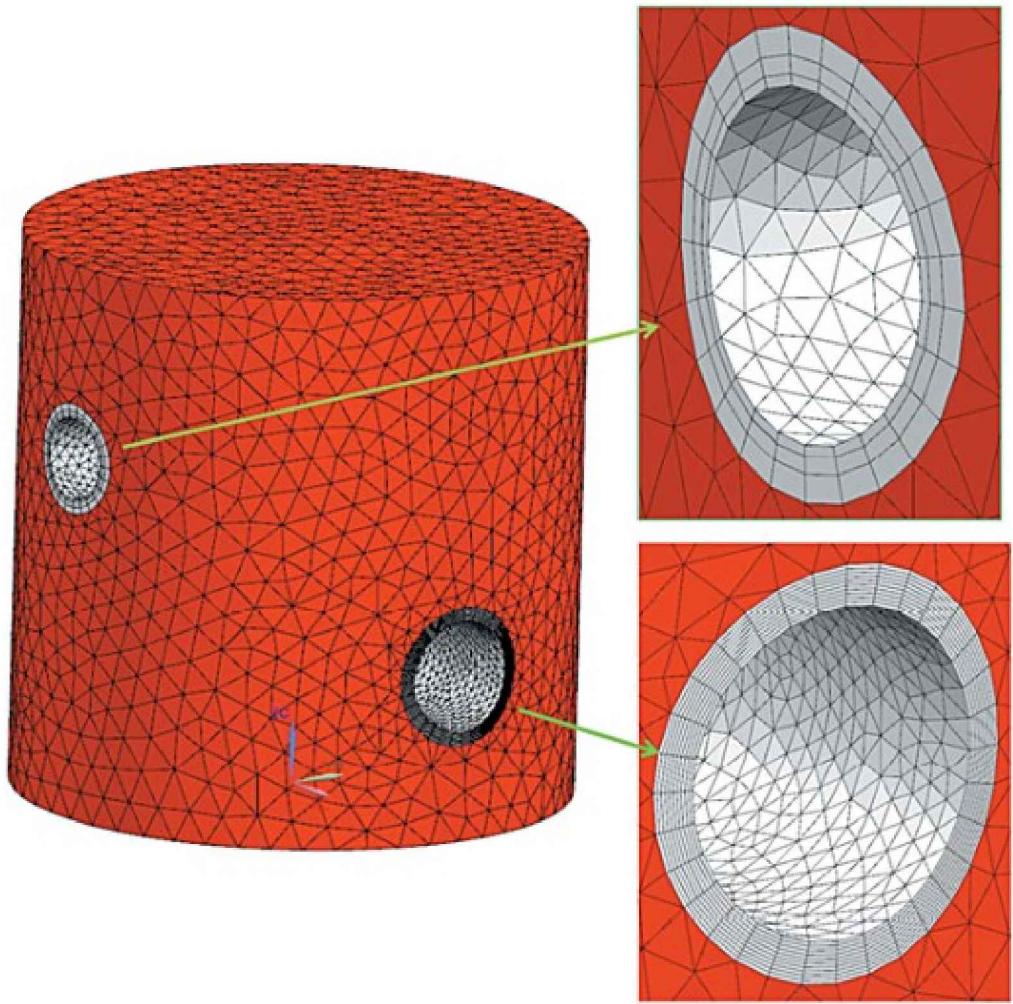


Figure 6.4. Results of the automatic mesh constructor with different wall-boundary mesh biasing parameters

6.4. Solution settings

One of the mandatory steps of formulating any solution involves selecting and setting up the corresponding mathematical models, that is, accounting for a set of physical processes. Depending on whether this selection is done correctly, the achieved solutions can deviate from each other and from the actual process. To account for the corresponding models in the NX Flow module, use options in the **Solve** dialog box as shown in Figure 6.5. To open this dialog box, choose the **Edit** command in the shortcut menu of the created Solution in the model tree. For a fluid dynamics problem, the **Solve** dialog box contains the following tabs and settings.

The *Solution Details* tab contains the following options:

- You can use the *Run Directory* list to specify the folder where the problem's files are located.
- You can use the *Solver Selection* list to select the solver type (for a detailed description, refer to para. 6.8).
- Under *Turbulence Model*, you can select the turbulence model or specify laminar flow. A description of turbulence models is provided in para. 6.7.
- Select the *Buoyancy* check box to account for buoyancy forces. To factor in these forces correctly, you need to specify ambient parameters, in particular, direction and magnitude of the gravity force. You can specify these parameters in the *Ambient Conditions* tab.
- Select the *Condensation/Evaporation* check box to model condensation or evaporation processes.
- Use the *Solution Type* list to select a steady-state solution (*Steady State*) or a solution that varies in time (*Transient*). If you switch to transient process mode, the *Transient Setup* provides the following additional settings:
 - *Advanced*: specify additional solution control parameters.
 - *Parallel Processing*: set up distributed multi-CPU computation (refer to para. 6.8).

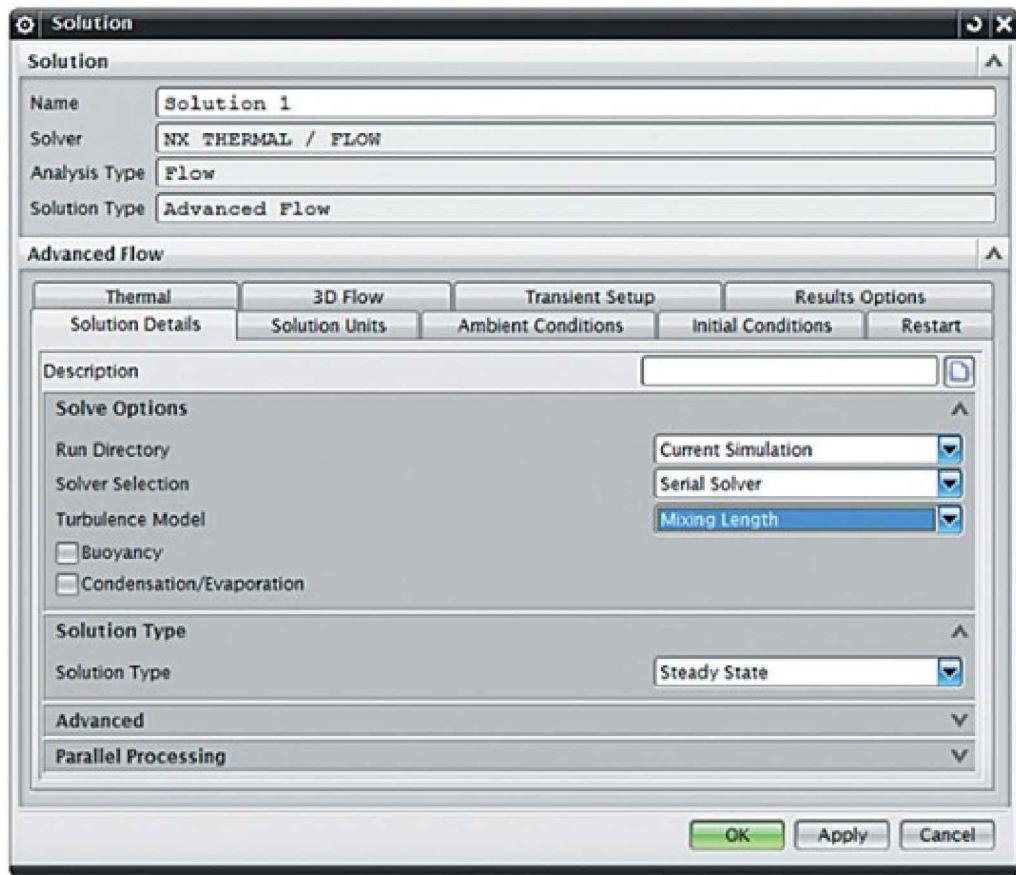


Figure 6.5. Solution parameters dialog box

You can use the *Transient Setup* tab to set up options of the problem for transient mode:

- *Start Time* and *End Time* options define the time interval at which the process is considered.
- *Time Integration Control* defines the time integration step.
- *Results Sampling* defines the saving frequency of solution results.

You can use the *Thermal* tab to account for compressibility of the flow in 1D Duct elements. To make it available, select the *Compressible Flow* check box.

You can use the *3D Flow* tab to set up the following parameters:

- The *Friction and Convection Parameters* are defined for all surfaces by default. You can override these using boundary

conditions.

- *Characteristic Scales* define the linear scale of flow that is used for the corresponding turbulence models.
- In the *Additional Parameters* group, the *Connect Disjoint Meshes* parameter activates stitching of disjoint meshes.
- *High Speed Flow* activates supersonic flows.
- *Non-Newtonian Flow* defines and configures models of non-Newtonian fluid models.
- You can use the *Fluid Domain Parameters* and *Fluid Domain Mesh Parameters* to set up the *Fluid Domain model and mesh*.

You can use the *Restart* tab to restart the simulation after it is stopped correctly.

You can use the *Solution Units* tab to choose units of measurement for parameters that are used in the problem.

You can use the *Result Options* to create a sample of data available for processing of results after the solution finishes.

You can use the *Initial Conditions* tab to specify initial fields of parameters for unsteady-state simulations.

6.5. Specifying boundary conditions

Correctly specifying boundary conditions is the most important component of problem formulation. The choice of mathematical model together with boundary conditions applied to it largely determines the adequacy of the obtained results.

You can use three lists of commands on the **Advanced Simulation** toolbar to specify initial conditions, boundary conditions, and specialized constraints:

- ***Simulation Object Type***
- ***Load Type***
- ***Constraint Type***

The ***Simulation Object Type*** list contains the basic boundary condition types (except thermal loads) that can be set using the ***Load Type***. You can use ***Constraint Type*** commands to set up initial conditions and temperature

fields, and to specify regions for data transfer between different solutions (*Mapping*).



Figure 6.6. *Simulation Object Type* command list

The following is a description of all commands in the **Simulation Object Type** drop-down list that can be used to specify basic types of boundary conditions (Figure 6.6).

The **Deactivation Set** command can be used to exclude selected elements of the model from the simulation process (this command works on all element types):

- *Eliminate from Solution* eliminates the selected elements from all calculation types.
- *Eliminate from 3D Flow Solution* turns off 3D flow simulation in the selected area.
- *Do not Convect* turns off convective heat exchange for selected objects.

The **Duct Flow Boundary Conditions** command can be used to set up flow parameters in one-dimensional ducts. For a complete description of this tool, refer to para. 5.5. In addition to the tools described in para. 5.5, there is a **Duct to 3D Flow Connection** connection type. This connection type links **1D Duct** elements and the 3D simulation region. It contains the following groups and parameters:

- You can use *Region* to define the surface where 1D and 3D elements are joined (data transfer interface from 1D to 3D).
- You can use *Flow Condition at Interface* to specify the method

used for coupling streams at the boundary. The default value, *Auto-Determine*, is recommended and it corresponds to automatically determining the coupling parameters. The other available values are: *Mass Flow* and *Pressure*.

- You can use *Head Loss* to account for head losses that occur at the coupling boundary.

You can use the **Flow Blockage** command to enter permeability parameters of the fluid. You can select one of the three permeability types:

- Select *Solid Blockage* to specify objects that are completely impermeable to the flow. This type is usually used to account for flow around bodies that participate in heat exchange.
- Select *Porous Blockage – Isotropic* to describe properties of an object with isotropic permeability properties that are different from the properties of the main flow.
- Select *Porous Blockage – Orthotropic* to describe materials that have different permeability properties in three mutually perpendicular directions.

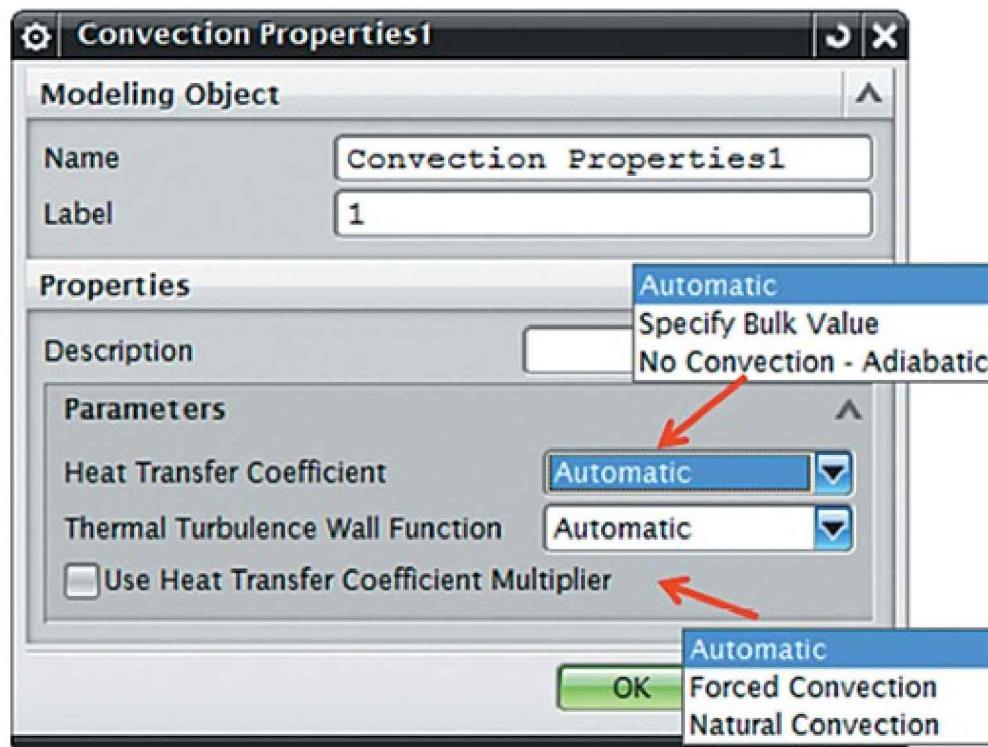


Figure 6.7. Specifying convective heat exchange properties

The *Solid Blockage* definition type has the following additional parameters:

- You can use *Wall Friction* to account for friction at the selected surface, and to describe properties of the flow surface. You can select one of the two options: *Smooth – With Friction* and *Rough*.
- You can use *Convection Properties* to set up parameters of convective heat exchange with the surface of the body using the **Modeling Objects** command (Figure 6.7).
- You can use *Slip Wall* to turn off sticking conditions on the wall and automatically activate the slip condition for that wall. Convective heat transfer from the wall is calculated from parameters specified in *Convection Properties*.
- You can use *Use Wall Function* to apply wall functions to the selected surface for the purposes of calculating friction and convection heat exchange at that wall. This option is applicable only to *K-Omega* or *SST* turbulence models (for a detailed description, refer to para. 6.7). If you clear this check box, low-Reynolds (Low-Re) modifications of *K-Omega* or *SST* models are used. In this case, conditions for using low-Reynolds models must be satisfied.

To describe convective heat exchange properties, you can use the *Convection Properties* object. In the **Convection Properties1** dialog box, the main parameter you can set is *Heat Transfer Coefficient*. Depending on the method you use to set up heat transfer coefficient, additional parameters become available.

The *Automatic* method of specifying the heat transfer coefficient (Figure 6.7) corresponds to calculating the coefficient based on fluid temperature data for the cell adjacent to the wall. In this case, the **Convection Properties1** dialog box contains the following parameters:

- In the *Thermal Turbulence Wall Function* list, you can select the method of accounting for turbulent flow parameters. If the flow regime is pronounced (free or forced convection), the value corresponding to the flow type is activated. If the flow regime is non-uniform or unknown, the *Automatic* option is used.

- The *Use Heat Transfer Coefficient Multiplier* check box is an extra parameter for entering the correction factor. It allows to account for aspects of heat exchange such as how developed the surface was when eliminated while simplifying the geometry, and aspects of process intensification.

If the heat transfer coefficient is known, the *Heat Transfer Coefficient* parameter value is switched to *Specify Bulk Value*. The temperature value of the fluid for heat transfer calculations is taken from *Reference Temperature For Bulk Heat Transfer Coefficients* in the *3D Flow* tab of the **Solve** dialog box (Figure 6.5).

If there is no heat exchange with the wall, the *No Convection – Adiabatic* value of the *Heat Transfer Coefficient* parameter becomes active.

The next command of the **Simulation Object Type** list (Figure 6.6) is **Flow Boundary Condition**. You can use this command to specify flow boundary conditions at ingress and egress of the simulation region, and to set up parameters of the internal fan model.

The *Inlet Flow* parameter in the *Type* group corresponds to ingress parameters of the simulation region. The following is the list of mandatory parameters:

- You can use the *Mode* parameter to specify the main ingress parameter. The available options are: *Velocity*, *Mass Flow*, *Volume Flow*, *Pressure Rise*, *Fan Curve*. The values can be immutable or they can vary with time and in space.

You can also use parameters in the following groups to specify additional data:

- In the *External Conditions* group, you can specify temperature, humidity, and turbulence parameters at the inlet boundary. By default, values from the *Ambient Conditions* tab of the **Solution** dialog box are used.
- In the *Heat Generated at Inlet* group, you can specify the heat that is imparted to the flow as it passes across the inlet boundary, or the heat that is withdrawn from the flow.
- In the *Tracer Fluid* group, you can introduce fluid particles into the

flow to evaluate their distribution and mixing. The fluid particles themselves do not influence the carrier flow in any way. The *Tracer Fluid* may not be used together with multi-component mixtures (*Homogenous Gas Mixture*). To select and create tracers, use the **Tracer Fluid List** command (Figure 6.8, 1). To activate the tracer fluid, set *External Conditions* that differ from the default conditions. The number of tracers to create is specified in parentheses in the *Tracer Fluid List* (Figure 6.8). The final stage of tracing the flow involves specifying the parts by weight of the different fractions. The sum of masses of the carrier flow and fractions must be equal to one. The mass of the carrier flow is calculated by subtracting the mass of all fractions from the total mass. To set up the fractions, enter values in the *Tracer Fluid Mass Fractions* box (Figure 6.8, 5). The order of mass concentrations corresponds to the order of the items in the *Tracer Fluid List* (Figure 6.8, 4). Tracer properties are defined by the *Molecular Diffusion Coefficient* and the *Turbulent Schmidt Number* (Figure 6.8, 3);

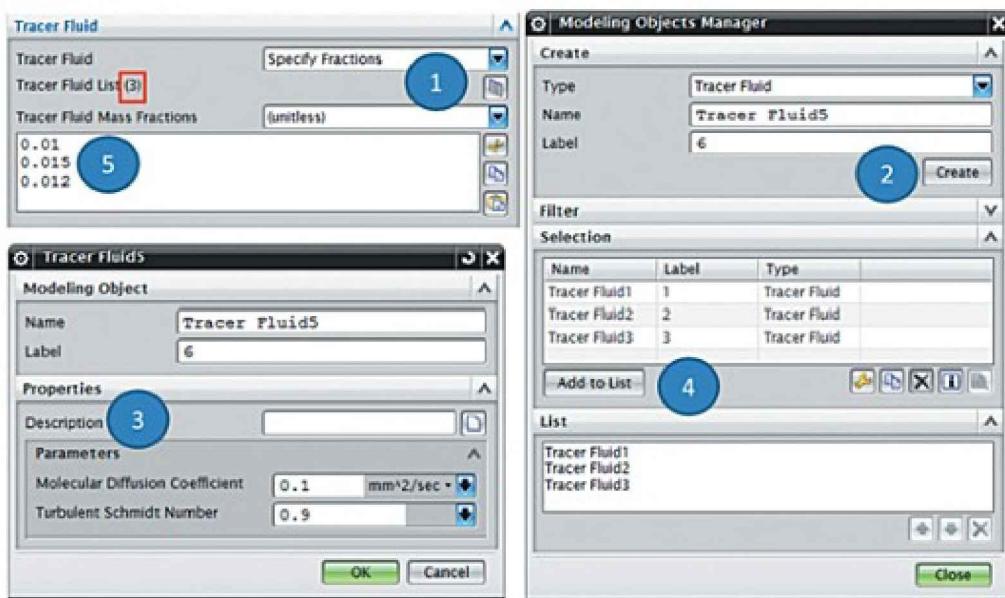


Figure 6.8. Creating carrier flow tracers

- In the *Mixture* group, you can set up modelling of multi-component homogeneous gas mixtures. The gas mixture is defined by

concentrations of components. To activate mixture modeling, set External Conditions that differ from the default conditions. To specify the composition of the mixture supplied into the simulation region, use the *Mixture* box (Figure 6.9, 3). To describe the gas mixture, create the corresponding ***Modeling Object*** called ***Homogeneous Gas Mixture*** (Figure 6.9, 2). This modeling object (Figure 6.9, 1) describes the set of gases that are mixed, the mixing parameters of the gases, the molecular diffusion coefficient, and the turbulent Schmidt number.

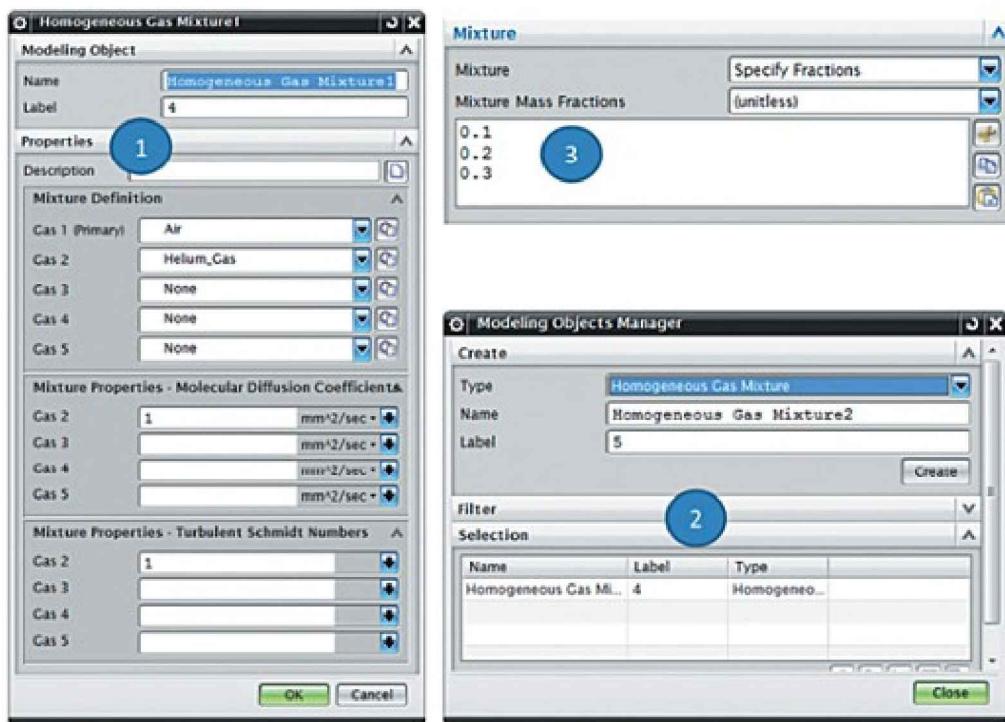


Figure 6.9. Specifying gas mixture flow

- In the *Flow Direction* group, you can specify the direction and swirl of gas at the inlet boundary. By default, gas is supplied along the normal to the surface and with no swirl. In the *Swirl* list, you can additionally specify the swirl of the flow: *Fixed Angle* or *Radially Varying*. If necessary, you can specify the injection angle across the boundary by selecting *Along Vector* in the *Alignment* list.

You can use the *Fixed Angle* option to specify the deflection angle (Figure 6.10, D) of the resultant vector (E) from the normal (C) in the plane formed

by the tangent (B) and normal (C) vectors. Vector (A) determines the flow direction. If this method is applied to all cells except the regions near walls, the velocity and direction of the flow are identical. The *Radially Varying* option specifies the injection of the flow rate component using the velocity value from the *Velocity* box, and the tangential component defined by the rotation speed and the distance from the centre of rotation $V = r\omega$.

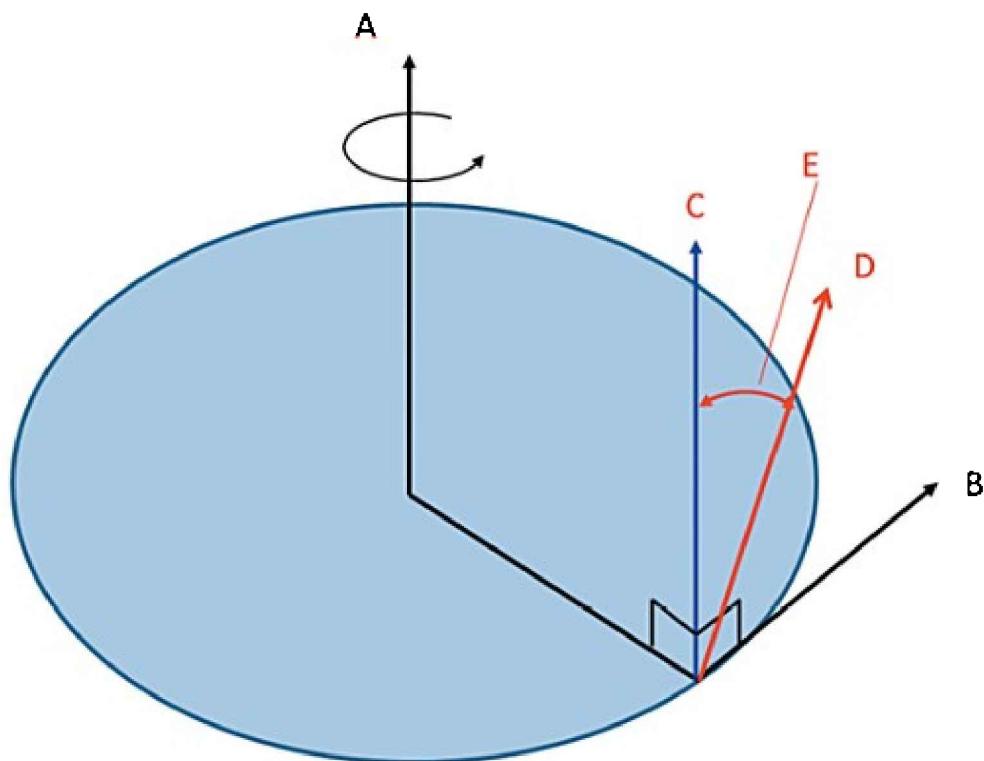


Figure 6.10. Flow swirl parameters at inlet

- In the *Fan Control* group, you can manage injection parameters based on information obtained from the simulation region. The *THERMOSTAT* option of the *Controller Type* allows you to create a control circuit that can be used to turn the flow across the boundary on and off whenever the corresponding on and off temperatures are reached. The *SPEED CONTROL* option allows you to adjust the rotation speed depending on the current temperature in accordance with the specified parameters. To enable both methods, install a *Temperature Sensor* and use its data to switch operating modes. If several sensors are installed, the average of

all temperature values is used.

- In the *Distribution* group, you can specify the spatial distribution of parameters at the boundary.

Depending on the selected *Mode* boundary condition, the set of groups and parameters in the **Flow Boundary Condition** dialog box partially changes. In the *Pressure Rise* mode, you can use the following additional groups and parameters:

- *External Absolute Pressure* in the *External Conditions* group.
- The *Head Loss* group for specifying local head losses. In this group, you can specify losses due to thin objects (perforated plates, screens, and so on). You can specify losses as a function of *Dynamic Pressure* or a function of *Approach Velocity*. You can also specify head loss by describing the shape of the object that causes the loss. The following objects can be considered: *Thin Perforated Plate*, *Metal Wire Screen*, *Silk Thread Screen*.

If as the *Mode*, you select *Fan Curve*, you can set up the head and flow rate relation of the device. The table method is used most often. The values of volumetric flow rate/pressure variables can be imported from an external file.

In addition to inlet conditions, in the **Flow Boundary Condition** dialog box, in the *Type* group you can add the following conditions to the region of interest.

The *Outlet Flow* boundary condition. You can use it to specify parameters at the outlet boundary. This condition is useful when the pressure/flow characteristic at the outlet of the simulation region is known. No flow can occur across this boundary towards the simulation region. This boundary condition is analogous to the *Inlet Flow* condition but the sign of the flow direction is opposite. Therefore, groups and parameters used with this type are analogous to the groups and parameters of inlet flow: *Fan Control*, *Head Loss*, *Fan Curve*, *Distribution*.

You can use the *Opening* boundary condition to specify a free inflow/outflow condition for the simulation region. The parameters are analogous to parameters of the *Inlet Flow* boundary condition. In addition, the *External Conditions* must be provided. The main parameter is the pressure of the

medium that receives the efflux. By default, static pressure, temperature, humidity, and turbulence parameters of the environment are taken from the values specified in the *Ambient Conditions* tab. In cases where parameters at the open boundary do not describe environmental conditions correctly, you can override them using *External Absolute Pressure* and *External Conditions*. You can use the *External Conditions* parameter to override temperature, humidity, and turbulence parameters. *External Absolute Pressure* is specified as an absolute value that must be greater than zero.

You can use the *Internal Fan* boundary condition to create an object that imitates a fan working within the simulation region. For this boundary condition, you need to specify the following parameters:

- Set up *Magnitude* to describe the pressure/flow characteristic. It is specified by selecting one of the following options in the *Mode* list: *Velocity* corresponds to the velocity of the fluid after the fan, *Mass Flow* and *Volume Flow* correspond to the flow rate of the fluid through the fan, *Fan Curve* corresponds to the head and flow rate characteristic.
- Enter *Heat Generated by Fan* to specify the heat that is uniformly transferred to the flow that passes through the fan.

The *Recirculation Loop* boundary condition describes the working fluid returning into the simulation region. Some work can be performed with the return flow. To set this boundary condition, specify the exit boundary of the simulation region (*Extract Region*) and the boundary across which the working fluid reenters the *Return Region*. For the boundary condition, specify the following parameters:

- Enter the *Magnitude* to specify flow rate characteristics of the flow that escapes the simulation region. The flow rates of the fluid escaping the region and the fluid re-entering the region are identical. You can specify the flow rate value in the same way as for the *Inlet Flow* boundary condition.
- Under *Flow Return*, specify the set of parameters that define the angle at which the flow enters the simulation area (along the normal or at an angle), rotation, turbulence parameters, and head loss. You can use the *Turbulence Characteristics* to specify the degree of tubulisation of the flow that re-enters the simulation

region.

- Under *Heat Exchange*, you can set up parameters to account for thermal processes that happen to the flow while it traverses the “invisible” part of the cycle, that is, before it enters the simulation region. As the influence acting on the flow, you can specify temperature change, or an amount of heat transfer. If the relevant parameters are known, you can account for convection heat exchange.
- *Distribution at Flow Return* is the description of the spatial non-uniformity of parameters at the boundary of the *Return Region*.
- *Flow Extract* allows to account for head loss at the outlet boundary if in the *Mode* list, you select *Fan Curve*. It is assumed that the losses are caused by objects whose thickness is much smaller than the other dimensions (screens, perforated plates).

You can describe properties of surfaces of objects immersed in the flow using the set of boundary conditions of the **Flow Surface** command. This type of boundary conditions is typically used to:

- Specify a slipping condition for the wall.
- Use wall functions of low-Reynolds turbulence models to transition to their high-Reynolds modifications.
- Specify wall roughness parameters.

This condition type is used to specify mandatory convection properties when solving thermal problems. In addition, you can specify surface motion parameters, which is especially important for modeling translating or rotating objects.

The **Flow Surface** command can work with two types of flow surfaces: double-sided and single-sided (or boundary) surfaces. Double-sided surfaces are located within the simulation region, that is, the fluid flows around these surfaces from two sides. The corresponding value in the *Type* group is *Embedded*. Single-sided (*Boundary*) surfaces are located at the boundary of the simulation area, so the fluid only flows along one side. Due to this subdivision, you need to specify properties for both sides of all double-sided surfaces. These sides are called the *Top Surface* and the *Bottom Surface*. For single-sided surfaces, only one side is defined.

Single-sided and double-sided surfaces are in turn subdivided into two types: *with Obstructions* and without. An obstruction is defined by the existence of certain small objects on the surface that are difficult to model by constructing a simulation mesh. These include, for example, small electronic components on a printed circuit board.

You can find out which side is the top surface and which is the bottom surface by visualizing the normal to the surface. In visualization, the normal vector is constructed from the top surface.

You can visualize and, if necessary, reverse the direction of the normal while working with the FE model by using the following command of the mesh collector shortcut menu: **Check All→Element Normals**. In the model verification dialog box, you can **Display Normals** and **Reverse Normals**. Remember that when you click **OK**, the action is committed, and the dialog box of the tool is closed. Clicking **Apply** commits the action but does not close the dialog box. Therefore, clicking **Apply** and then clicking **OK** causes a double reversal of the normal, which is thus returned to the initial state.

To use the **Flow Surface** command, specify the following parameters:

- Under *Slip Wall*, set up slipping. If you activate slipping for selected surfaces, roughness and turbulence settings are automatically made unavailable, and sticking condition is not applied.
- Under *Use Wall Function*, specify wall functions for low-Re turbulence models (SST and K-Omega). This option is not applicable for other turbulence models.
- Under *Wall Friction*, set up wall friction. This parameter is available if the *Slip Wall* check box is selected. You can use this parameter to specify roughness properties of the surface. The following values are available: *Smooth – With Friction* is the default option with no wall roughness. *Rough* sets up equivalent sand roughness using the *Sand Grain Roughness Height* parameter. *Rough – From Obstructions* specifies a roughness correction value, taking into account obstacles to free flow.
- Under *Convection Properties*, specify individual convection properties for selected surfaces. This option is implemented using the **Modeling Object** command.

- Under *Rotation/Translation Parameters*, specify the type of motion of the surface relative to the flow. The motion can be rotational or translational. In case of rotation, specify the axis and direction of rotation, a point on the rotation axis, and the rotation speed. In case of translation, specify the travel direction and velocity.

You can use the **Fluid Domain** command to automatically construct a simulation mesh in the fluid domain. This command supports two construction types. The first type constructs the mesh in the main fluid domain: *Fluid Mesh*. The second type constructs the mesh near the wall: *Fluid Surface Mesh*. The primary tool is the *Fluid Mesh*, and the secondary (optional) tool is the *Fluid Surface Mesh*.

If *Fluid Mesh* is selected, the list in the *Type* group contains the following basic parameters of the command:

- Under *Fluid Domain Region*, select the location of the flow zone where you need to construct a mesh. The mesh generator automatically finds boundaries of the region and excludes all bodies in the selected area that are not marked as fluid domains (for more information, refer to para. 6.3).
- Under *Inner Region to Mesh Through*, specify the regions where the mesh must not be constructed.
- Under *Material*, specify physical properties of the fluid that occupies the fluid domain. The fluid can be a *Pure Substance* or a *gas Mixture*. You can select the substance in the materials library. You can specify the gas mixture by setting up a modeling object.
- Under *Mesh Density*, specify characteristic sizes of cells of the simulation mesh. You can enter the cell size in relative units (cell size relative to the longest edge of the region) or alternatively you can specify the absolute size of a cell edge.

If in the *Type* group, you select the *Fluid Surface Mesh* option, you can specify data for building the mesh that is located at the surface of bodies and in the region adjacent to the bodies. In this case, the prismatic mesh is built in accordance with the following parameters:

- Under *Region*, select surfaces for building the surface and/or wall mesh.

- Under *Surface Mesh density*, specify characteristic dimensions of the surface mesh. To make it available, select the *Specify Local Surface Mesh Density* check box. If this check box is not selected, the surface mesh is constructed in accordance with the volume mesh that was created using the *Fluid Mesh* option in the *Type* group list. The cell size can be specified as an absolute value or relative to the longest edge of the relevant region. You can also use the template mesh of the FE model as the surface mesh.
- *Boundary Layer Mesh Definition* builds a prismatic simulation mesh adapted to the wall. A wall FE mesh is used for higher-precision flow modelling in the boundary layer, which is particularly important for low-Re turbulence models. To make this parameter available, select the *Create Boundary Layer Mesh* check box. You can specify the mesh for this option using two methods: *Total Thickness* and *Relative Thickness Ratio*. In the first case, the thickness of layers is determined by the following parameters: *First Layer Thickness*, *Total Thickness* (thickness of the whole wall mesh), *Number of Layers*. In the second case, you can specify a layer biasing coefficient (*Thickness Ratio*) and either *Number of Layers* or the thickness of the first layer (*Specify First Layer*). The number of layers is then chosen automatically based on the specified layer biasing coefficient and the characteristic cell size in the fluid domain.

You can use the ***Mixing Plane*** command of the ***Simulation Object Type*** set (Figure 6.6) to set up a virtual boundary between flow zones with different parameters. The main purpose of this boundary condition is to represent the flow as an integral (average) characteristic of the flow that passes from one region to another. This type of boundary condition is particularly useful for modeling rotating (rotor) and stationary (stator) stages of a bladed machine, when the flow can be represented as an averaged value of velocity and pressure. The averaging takes place at the outlet of one of the regions and its value is passed to the adjacent simulation region. There are two types of mixing planes:

- *Mixing Plane with Joint Mesh Interface* is a plane that connects regions that are mated node-to-node in the mesh model.
- *Mixing Plane with Disjoint Mesh Interface* is a mixing plane that

connects disjoint mesh models.

In the first case, the model must be described by a single region, while in the second case, the model can contain two or more regions. In both cases, you need to specify the direction of the flow and parameters of flow averaging at the mixing boundary. However, for node-to-node mating (*Joint Mesh*), you need to specify an internal mixing boundary, while for connecting disjoint regions (*Disjoint Mesh*), you need to specify a downstream boundary and a connected upstream boundary (*Upstream Region* and *Downstream Region*). Both options employ identical methods to average parameters at the connection boundary: along radius and along a direction. The *Along Radius* averaging method is typically used to connect cylindrical regions. In other cases averaging is typically performed *Along Vector*. The mating boundary is additionally split into the specified *Number of Segments*. The averaging is performed for these segments.

You can use the **Moving Frame of Reference** command to create a moving coordinate system and introduce terms that describe Coriolis forces and centripetal acceleration into the system of equations. You can use this command to model processes where you need to account for rotation or translation of objects. Rotating coordinate systems are typically used in modeling bladed machines (fans, pumps, compressors). Translation of coordinate systems allows you to account for motion under variable external loads, for example, oscillation of fluid in a tank under external force. You can use the following data to define rotating coordinate systems (*Rotating Frame of Reference*):

- *Region* specifies the rotating object.
- *Rotation Parameters* specify the direction and speed of rotation.
The direction is specified by *Specify Vector* and *Specify Point*.

For a translating coordinate system, in the *Type* group, select *Translating Frame of Reference* and specify the following data:

- Under *Region*, specify the moving object.
- Under *Translation Parameters*, specify the acceleration value of the object in the relevant directions. In contrast to rotating systems, acceleration can be specified as a function of time. If necessary, you can specify scale factors for accelerations in the

corresponding directions Ax, Ay, Az.

You can use the **Particle Injection** command to create objects that describe injection of particles into the carrier flow. To define the particles, specify the injection point and physical properties of the particles. The motion of particles in the simulation region depends on the flow parameters and on physical properties of the particles. The particles themselves cannot influence the carrier flow. The concentration of particles is specified either as the number of particles per unit volume of gas, or as a number of particles injected per unit of time. The particles can be treated as massive or massless objects.

For particles that have neither mass nor shape, specify the following parameters:

- Under *Inflow Density*, select the method used to specify the concentration of particles. You can select either *Number per Volume* or *Number per Second*.

If the *Mass and Size Options* check box is selected, you can additionally specify the following:

- *Mass*, which defines inertial properties of the particles.
- *Diameter* of the particles.
- *Drag*, which defines the shape of the particles. Two values can be specified: *Spherical Correlation* for spherical particles and *Specify* for cases with non-sphericity.

In addition, more values of the *Inflow Density* particle concentration parameter become available:

- *Mass per Volume*.
- *Mass per Time*.

If for the *Inflow Density* parameter, you select *Number per Seconds*, you also need to specify the injection velocity of the particles. The velocity can be equal to the flow velocity (by default) or it can have a different value.

You can use the **Periodic Boundary Condition** command to create a simulation model that accounts for periodic repetition of the processes,

which can significantly reduce the dimensionality of the problem. A necessary but not sufficient prerequisite for correctly applying this boundary condition is periodicity of geometry. There are two types of periodicity in NX Flow: rotation and linear. *Rotation Periodicity* is used to describe models that can be considered a sector that is repeated with a certain increment in a cylindrical coordinate system. To describe periodicity of this type, specify the rotation axis of the model and the boundaries of the repeated sector. If the model can be considered as linearly repetitive, use the *Translational Periodicity* type. In this case, you only need to specify boundaries of the periodicity region. You can additionally impose a *Pressure Drop* condition when the flow passes the periodic section.

Periodic boundary conditions can be used if the model satisfies the following prerequisites:

- Periodicity boundaries must be geometrically identical.
- Simulation meshes at periodicity boundaries must be completely identical.
- Both faces describing the periodicity region must be specified as periodicity boundaries.

You can use the **Report** command for reporting on various integral and local properties of the flow. There are four report types:

- *Per Region* provides data at the boundary of interest.
- *Between Regions* provides data between boundaries.
- *Track During Solve* provides data during the solution process.
- *Lift and Drag* provides data of the lift and drag forces.

The following is a description of the features each of the reports delivers.

- *Per Region* provides flow data for a selected boundary. You can use the *Velocity and Pressure*, *Temperature*, and *Turbulence Data* parameters to obtain information on the minimum, maximum, and mean values of the corresponding parameters. You can use the *Velocity Gamma Values* parameter to obtain information on the non-uniformity of velocity at the selected boundary. A value of one corresponds to complete uniformity of the flow. *Mass and Volume Flow* provides information on mass and volume flow rate. This

information is integral (summed up at boundaries of all cells in the selected region). *Report Data on Specified X-Y Plane* reflects all selected results for a given plane. *Convection to Fluid* provides information on convection from the selected surface to the environment.

- *Between Regions* provides data on the flow between two specified boundaries. You can select *Pressure Drop* and *Spectral Attenuation* parameters. You can use the pressure drop to determine the drop of static and full pressure between two specified boundaries. Spectral attenuation provides information on unsteady-state attenuation of acoustic fields based on the existing pressure field.
- *Track During Solve* provides information on the primary flow parameters (velocity, pressure, temperature) for the selected object (element, region) during the simulation process. Monitoring results are added to the *Track Results* tab of the **Solution Monitor**.
- *Lift and Drag* provides data on integral parameters of the immersed object, such as the lift force and drag. You can use this report to calculate the aerodynamic properties of the selected object. To obtain the corresponding Cx, Cy, Cz coefficients, select the *Calculate Coefficients* check box and specify geometry, velocity, and physical parameters of the model.

You can use the **Screen** command, which creates a two-dimensional object, to describe local flow resistance. You can specify resistance parameters using the *Planar Head Loss* modeling object. For more details, refer to para. 6.6.

You can use the **Selective Results** command to selectively process computation results in the specified region. Use this command to select regions where (and only where) results will be produced for the data types of interest.

You can use the **Supersonic Inlet** command to specify supersonic flow parameters at the inlet of the simulation region. You can specify the following parameters:

- *Mach Number*, which is the main property of supersonic flow.

- *Inlet Conditions*, that is, static pressure and temperature values. By default, the values are equal to atmospheric parameters specified in the *Ambient Condition* tab of the **Solution** dialog box. If necessary, in the *Inlet Conditions* list, you can override the conditions with the *Specify* option.
- Under *Flow Direction*, you can specify the flow direction relative to the boundary. By default, the inflow is normal to the specified surface. If necessary, you can change the flow direction with the *Specify* option.
- Under *Turbulence at Inlet*, you can specify turbulence properties of the flow. In this boundary condition, you can define turbulent flow through degree of turbulization and scale of the eddy (*Turbulent Intensity and Eddy Length*), kinetic energy of turbulent pulses and dissipation of turbulent energy (*Turbulent Kinetic Energy and Dissipation Rate*), or kinetic energy of turbulent pulses and dissipation rate of turbulent energy (*Turbulent Kinetic Energy and Specific Dissipation Rate*).

You can use the **Symmetry Plane** command to specify the plane at which adiabaticity, impermeability, and frictionlessness conditions are satisfied. Velocity vectors are always directed along the symmetry plane, while all flows orthogonal to the plane are equal to zero. This condition is typically applied to reduce the dimensionality of the problem.

You can use the **Thermal Coupling – Convection** problem to define the thermal coupling that imitates convection heat exchange. This boundary condition introduces simplified calculation elements into the model using 1D ducts. It is useful in cases where you have data on the flow being studied but for whatever reason it is impractical to use the classical-formulation modeling based on solving the Navier-Stokes equations. Additional information on element coupling parameters is provided in Chapter 5 of Part 2.

The **Load Type** command is used for specifying thermal load data. This tool is described in para. 5.3.4.

Constraint Type commands are used to specify the initial distribution of flow parameters (*Initial Conditions*), specify zones for data exchange with other analysis type (*Mapping*), and specify temperature (*Temperature*). For

a detailed description of these tools, refer to para. 5.3.

The **Initial Conditions** command accepts additional parameters typically encountered in fluid dynamics problems, such as initial pressure fields, velocities, humidity values, turbulence properties, and so on.

To transfer data regarding the force that the flow exerts on the immersed structure using the **Mapping** command, select the surfaces whose data need to be collected, in the initial fluid dynamics model. In the strength model, specify the corresponding surfaces to which these loads need to be applied.

6.6. Modeling pressure losses

NX Flow supports solving a broad range of problems, including problems that involve fluids flowing through objects of limited permeability. These objects include perforated plates, filters, screens, and many other elements of flow channels. However, in cases where the flow through such an object is not the ultimate unknown of the problem, or if the dimensionality of the resulting problem makes it unsolvable, specialized engineering approaches are more effective. One of these approaches in NX Flow involves using the *Planar Head Loss* modeling object. This object accounts for head losses due to the fluid passing through a thin permeable obstacle.

The following main parameters are used: obstacle type, method of loss calculation, and the value of the loss coefficient. The following obstacle types are available:

- *Specify* corresponds to an unspecified obstacle with a known loss coefficient value. You can specify the head loss coefficient or the pressure loss proportionality coefficient with regard to velocity.

In the first case, the standard loss coefficient equation is used, and the differential pressure is $\Delta P = k_{loss} * V^2 / (2g)$ where k_{loss} is the specified head loss coefficient, g is the free-fall acceleration, and V is the velocity of the flow.

If you use the proportionality coefficient, a function f is introduced, which defines the relation of differential pressure and fluid velocity as

$$\Delta P = -fV$$

- *Thin Perforated Plate* is the tool that allows to calculate the head loss coefficient from specified plate perforation parameters. To describe the plate, select a type to represent *Orifice Geometry*. The edges of the holes can be rounded (*Rounded Edge*), sharp (*Sharp Edge*), or conical (*Beveled Edge*). For sharp edges (Figure 6.11, A), only the ratio of the internal cross-section area to total cross-section area (F_{open}/F_{total}) is specified as the *Free Aspect Ratio*. For rounded edges (Figure 6.11, B), you can specify the rounding radius (*Edge Radius*), the diameter of the internal cross-section (*Orifice Hydraulic Diameter*), and the ratio of internal cross-section area to total cross-section area (*Free Aspect Ratio*). For beveled edges (Figure 6.11, C), you can specify the same parameters and an additional *Plate Thickness parameter*. After that, in all cases, you need to calculate the loss coefficient. To do so, click **Compute**.
- *Metal Wire Screen* describes flow through a screen with fine metal mesh. For a *Metal Wire Screen* obstacle, specify the ratio of internal cross section area to total cross section area (*Free Aspect Ratio*).
- *Silk Thread Screen* describes flow through a screen with a fine non-metallic mesh. For the non-metallic mesh obstacle (*Silk Thread Screen*), just as for a metallic one, you only need to specify the ratio of internal cross section area to total cross section area (*Free Aspect Ratio*). The difference between metallic and non-metallic meshes lies in the corresponding correlation ratios that are used for calculating loss coefficients.

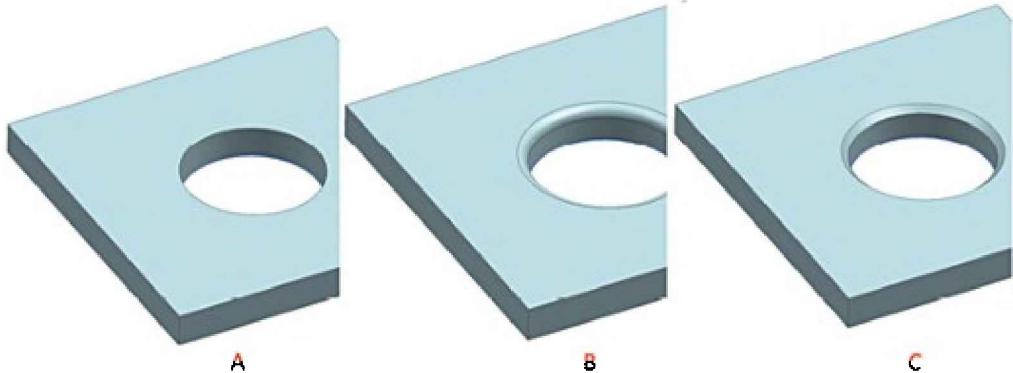


Figure 6.11. Perforated plate edge types: A. Sharp edge. B. Rounded edge. C. Bevelled edge

6.7. Turbulence models

You can use NX Advanced Flow for numerical modeling of various flow classes. You can use a variety of criteria to classify flow types, particularly, the degree of turbulization. With regard to this criterion, flow is typically described as laminar, turbulent, or transition flow. The most important criterion that describes the flow regime turbulence-wise is the dimensionless ratio of inertial forces to viscous forces, called the Reynolds number (Re). Historically, the most often used value of Re that corresponds to laminar flow turning turbulent is 2300. It must be noted that this value is applicable only to O. Reynolds' experiment with flow in cylindrical tubes, in which this transition point value was obtained.

Modeling of turbulent flow in NX Advanced Flow is based on applying the so-called Reynolds approach to describing flow. This approach involves representing any instantaneous values of fluid-dynamics parameters as a sum of the time-averaged value and its pulse components. This approach is called Reynolds averaging, and the solution model is called the Reynolds-averaged three-dimensional unsteady-state Navier-Stokes equation model.

NX Advanced Flow implements three alternative models of fluid flow that allow accounting for turbulence:

- *The None – Laminar Flow* model does not take into account turbulent pulse generation. This model is applicable only if the Reynolds number is low (typically, for $Re < 2000$).

- The *Fixed Turbulent Viscosity* model describes turbulent viscosity using the following relation:

$$\mu_t = 0.01 \rho V_m l ,$$

where V_m is the average velocity value.

l is the characteristic scale of the turbulent eddy.

- *Mixing Length*. This model is based on the assumption that there is a certain characteristic linear dimension l along which a turbulent mole (a finite volume of fluid) retains its individuality. In this case, turbulent viscosity is represented as

$$\mu_t = \rho l^2 S ,$$

where l is the mixing length scale.

S is a function of the simulation point location: near a wall or in the free flow zone.

- *K-Epsilon* model. One of the most often-used turbulence models. It belongs to the class of two-parameter models. NX Advanced Flow uses an implementation of the $k-\varepsilon$ model that is a standard (high-Re) model proposed by Launder and Spalding in 1972. In this implementation, turbulent viscosity is calculated using the following expression:

$$\mu_t = \rho C_\mu k^2 / \varepsilon ,$$

where ε stands for turbulence dissipation, k is the turbulent kinetic energy.

It is commonly accepted that in high-Re modifications of models, turbulent flow near walls is described with the corresponding wall function. Using such a function allows the formulation of boundary conditions for equations of velocity and turbulent properties (k and ε) along the wall, because the high-Re modification of the model itself is not well adapted for this. In NX Flow, selecting a $k-\varepsilon$ model implies using a high-Re model with wall functions. Thus, there are certain limitations with regard to mesh cell size near walls. The limitation can be formalized as the range of values of dimensionless distance between the wall and the top face of the wall-

boundary cell, which is signified by y^+ . The $30 < y^+ < 100$ range is considered the working range. In this case, the function that describes flow near the wall falls within the logarithmic zone of the boundary layer (Figure 6.12, 3). If you construct a mesh with $y^+ < 30$, there is a probability of hitting the transition or even viscous zone of the boundary layer, which leads to significant loss of precision due to forced reduction of the layer's thickness (Figure 6.12, 1 and 2). If the y^+ value is significantly greater than 100, there is a probability of increasing the thickness of the boundary layer, again causing computation errors (Figure 6.12, 4 and 5).

- The *K-Omega* model. NX Advanced Flow uses an implementation of the Wilcox $k-\omega$ model (1998), which calculates turbulent viscosity using the equation:

$$\mu_t = \rho k / \omega,$$

where k is the turbulent kinetic energy, ω is specific dissipation.

This model is formulated as a low-Re model, but it can be switched to using wall functions (to reduce computational intensity in areas where using this technique is acceptable). For such a switch, you can use the ***Flow Surface*** boundary condition and specify surfaces where wall functions need to be applied. You can then select the *Use Wall Function* check box. In this case, all selected surfaces use the analog of the high-Re modification of the model (that is, $k-\omega$ models with wall functions). If you specify this parameter, rules formulated for the high-Re $k-\epsilon$ model are applied to the mesh model ($30 < y^+ < 100$). If necessary, you can enforce wall functions on all surfaces by selecting the *Use Wall Function* check box in the ***3D Flow*** tab of the ***Solution*** dialog box. In all other cases, the y^+ criterion parameters must satisfy the requirements of low-Re turbulence models:

- a. $y^+ < 5$ (even better, $y^+ < 1$): in this case at least one cell ends up in the viscous sub-layer (if $y^+ < 1$, at least 3–4 cells end up in the viscous sub-layer).
 - b. Mesh biasing parameters (ratio of adjacent steps) must be under 1.3, that is, the cells should get larger gradually.
- *Shear Stress Transport – SST* corresponds to the Menter model. It is a hybrid model built upon the $k-\omega$ model. In this case, the $k-\omega$

model is used in the vicinity of the wall, and the $k-\varepsilon$ model is used in the external flow. The reasoning behind this model is that $k-\varepsilon$ models are well-suited to describing free shear flows, while $k-\omega$ produce good results in modelling flow in the vicinity of walls. The main difficulty lies in correctly determining the point of switching from one model to the other. NX Advanced Flow implements the most advanced modification of the SST model that represents turbulent viscosity as $\mu_t = \rho a_1 k / \max(a_1 \omega, \Omega F_2)$ (based on Bradshaw assumptions and modifications for flows bounded by walls where the Bradshaw assumption is not necessarily applicable [14, 15, 16]). Since in the vicinity of the wall, this model uses the $k-\omega$ implementation, the applicable simulation mesh construction principles are the same as for the $k-\omega$ model (that is, initial low-Re modification and ability to use wall functions).

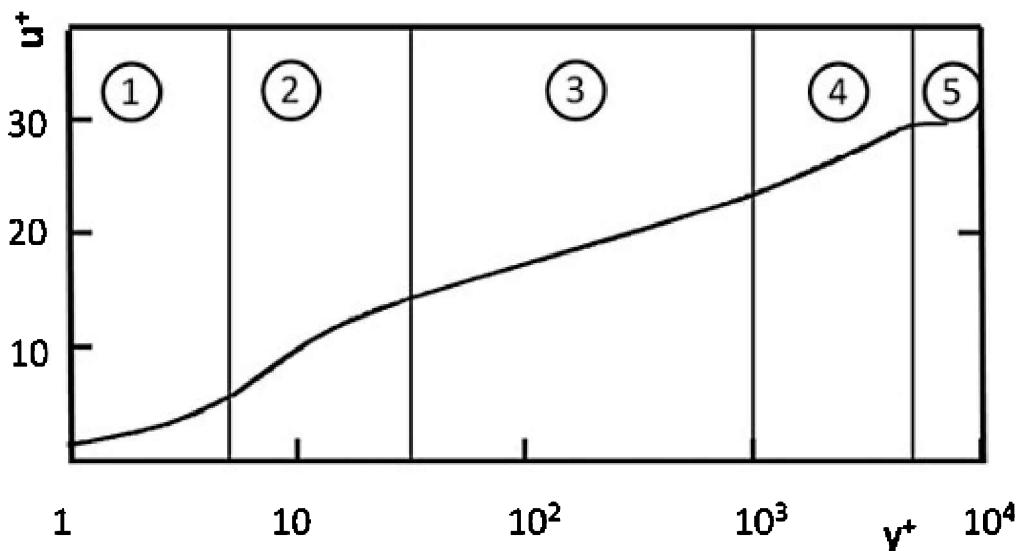


Figure 6.12. Velocity profile of turbulent boundary layer, log scale: 1. Viscous sub-layer. 2. Transition zone. 3. Logarithmic zone. 4. Velocity defect zone. 5. Intermittency area

The graph in Figure 6.12 is shown in terms of wall law variables. Here $y^+ = yv^*/\nu$, $u^* = u/v^*$.

6.8. Parallel computing in NX Advanced Flow

At present, one of the most important routes to increasing calculation speed

and precision (by using finer mesh models) involves using multithreaded solvers, that is, parallel computing. The parallel computing technology must be supported by hardware and software. Availability of a parallel solver is a necessary but insufficient prerequisite of parallel computing. The following is a description of settings that you need to configure to use parallel computing on a PC with a Windows operating system and a multicore CPU.

The MPI (Message-Passing Interface) protocol is used for communication between CPUs or cores of the same CPU. NX Advanced Flow uses the MPICH2 implementation of the MPI protocol, which is freely downloadable from the official website (<http://www-unix.mcs.anl.gov/mpi/mpich/>).

To make full use of parallel computing features, install and configure MPICH2. After that, you can configure NX Advanced Flow.

To solve an NX Advanced Flow solution in parallel mode, open the **Solution** dialog box, select the **Solution Details** tab, and in the **Solver Selection** list, select **Parallel Solver**. After that, in the **Parallel Processing** group, select the **Run Solution in Parallel** check box. The parallel computing setup mode activates. All settings of this computing mode are stored in the **Parallel Configuration File.xml**. If you have a prepared parallel configuration file, specify it in the corresponding box using the **Browse** button.

If you do not have this file or if it needs editing, use the **Parallel Configuration Tool** that you can call using the **Create New File** or **Modify Existing File** commands respectively. When you run this tool, the system makes sure that the MPI protocol is installed correctly. If no errors are found, a Success status message is displayed (Figure 6.13). Further settings of the parallel mode are similar both for the Advanced Flow module and for the Advanced Thermal module. You can select the module to configure in the **Computer Lists** group, in the **View Factors** (for the Thermal module) and **Flow** (for the Flow module) tabs, as shown in Figure 6.13.

After selecting the **Flow** tab of the **Parallel Configuration Tool** dialog box, you can set up the number of threads to split the problem into, in the **Settings** group. You can specify the maximum number of available CPUs/cores in the **Number of processors on computer** box, and you can specify the number of CPUs/cores assigned to the simulation in the **Number of processors to use** box. You can manage the degree to which the solution

is parallelized by varying the number of CPUs/cores assigned to the solution. After making all necessary settings, save the settings file by choosing **File→Save As...**. After saving, check the contents of the saved *Parallel Configuration File.xml*. The most important value to verify is the *Number Of Processors* in the *Flow Computer List* property block. This number you see must be the same number you specified. If necessary, change this value and save the file again.

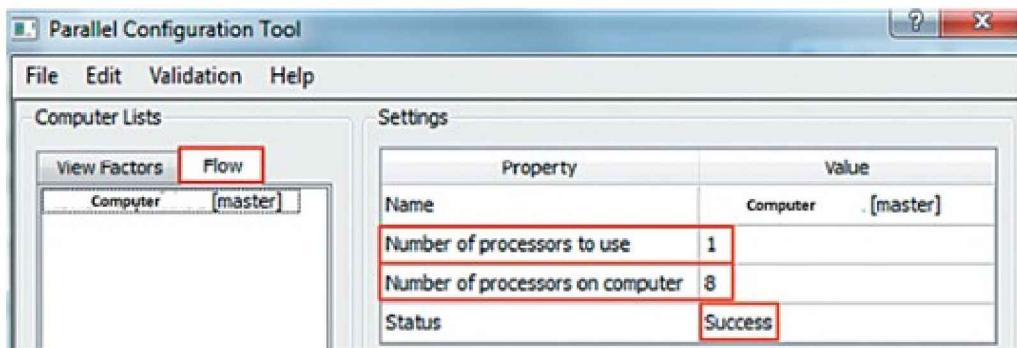


Figure 6.13. Configuring parallel computation

After preparing a settings file, select it in the *Parallel Configuration File* box.

Note that some types of objects and boundary conditions are incompatible with parallel computing. If any incompatibility is found, the system signals an error when the simulation is run.

6.9. Example solutions of problems

6.9.1. Modeling free-convection flow in the gap between two coaxial cylinders

This problem analyzes the free-convection flow that arises in the gap between two infinitely long coaxial cylinders that have different temperatures. All external (with regard to the fluid domain) boundaries are insulated. Radiant heat exchange is neglected. Solution results are compared to experimental data [17, 18]. The original geometry of the model is shown in Figure 6.14, A. The diameter of the outer cylinder D_o is 92.6 mm, and the diameter of the inner cylinder D_i is 35.6 mm. Temperature T_1 is 373 K, and T_0 is 327 K.

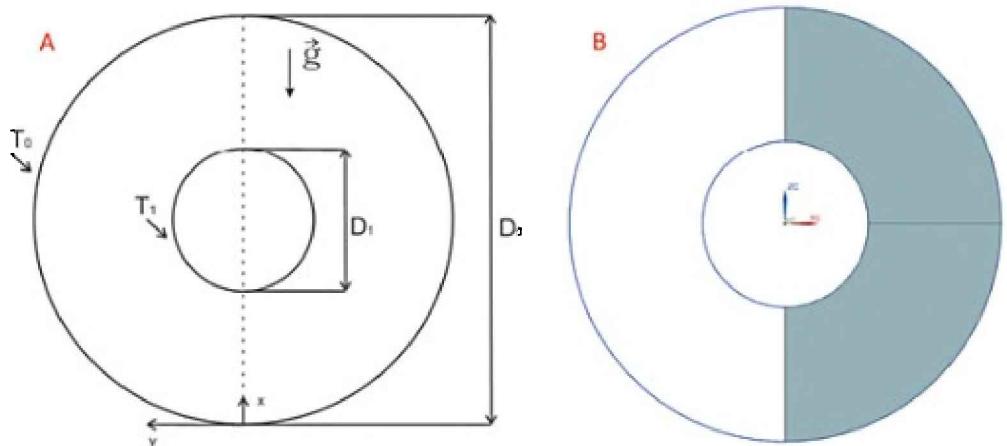


Figure 6.14. Original geometry model of the gap

The problem will be modeled using symmetry. The plane of symmetry runs along the direction of the gravity force and passes through the centre of cylinders. To simplify the subsequent construction of the simulation mesh, the model is divided into two equal parts (Figure 6.14, B).

To solve the problem in NX Flow, you need to construct a simulation mesh using an NX Thermal/Flow template (Figure 6.15).

Name	Type	Units	Relationship	Owner
NX Nastran	Fem	Millimete...	Stand-alone	NT AUTH...
NX Thermal/Flow	Fem	Millimete...	Stand-alone	NT AUTH...

Figure 6.15. Selecting a simulation mesh template

To use this template, right-click the name of the model in the **Simulation File View** dialog box and choose **New FEM** in the shortcut menu. In the **FEM** template selection dialog box, specify the *NX Thermal/Flow* template (Figure 6.15), and make sure that the working *Folder* and the *Name of the mesh model file* are set up correctly. Click **OK** to confirm. Note that the **Simulation File View** now contains a new *model_name_fem* node that corresponds to the *FEM* file.

Since geometry preparation involves splitting the solid-body model in two,

specify mating conditions for nodes of FE meshes. To do so, use the ***Mesh Mating Condition*** command on the **Advanced Simulation** toolbar. Select all bodies to mate them with each other. As the mating *Type*, select *Automatic Creation*. Keep default values for the rest of the settings (Figure 6.16).

A structured FE mesh adapted to walls is then created for the problem. Creating such a mesh involves using the ***Mesh Control*** tool. Specify the following splitting parameters for radial edges (Figure 6.17):

- *Density Types – Biasing on Edge*.
- *Bias Origin – Center of Edge*.
- *Number of Elements* – 50.
- *Bias Ratio* – 1.1.

For circular edges (Figure 6.17), specify uniform splitting (*Number on Edge*) with a value of 30.

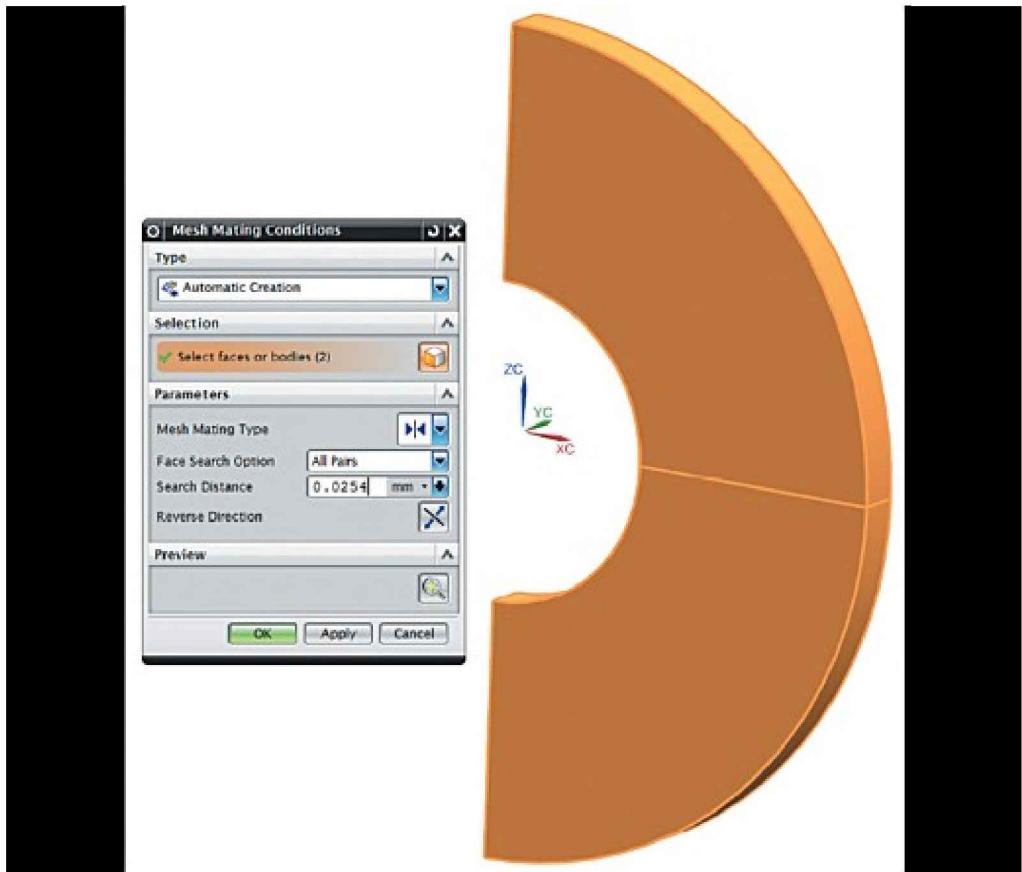


Figure 6.16. Stitching model parts to construct a simulation mesh

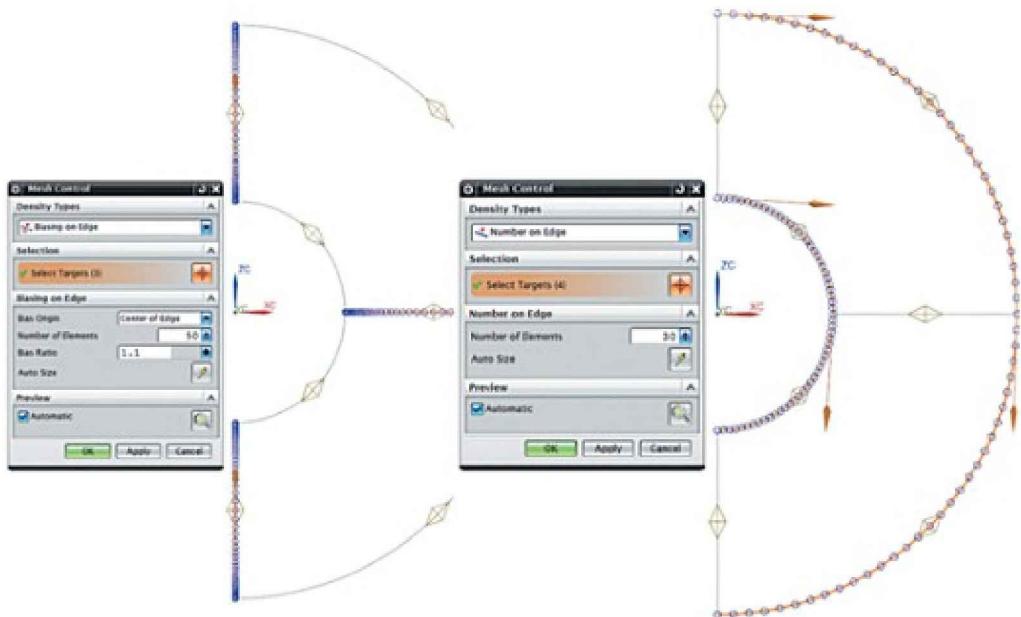


Figure 6.17. Specifying splitting parameters

Construct a template 2D mesh to create an ordered 3D hexahedral mesh. To construct a 2D mesh, use the **2D Mapped Mesh** command on the **Advanced Simulation** toolbar. The auxiliary mesh is built on the faces where splitting parameters are defined. Select *QUAD4 Thin Shell* as the element type. Clear the *Export Mesh to Solver* check box to exclude 2D mesh elements from the simulation. Click **OK** to complete the mesh construction process (Figure 6.18, A).

Surfaces of the auxiliary mesh are then used to construct the 3D mesh that is one element thick (for modeling flow that is symmetric with regard to the cross section of the cylinders). To construct this mesh, use the **3D Swept Mesh** command. As the type, select either *Until Target* or *Multi Body-Until Target*. Two 3D meshes are constructed regardless of the type. In the **3D Swept Mesh** dialog box, specify the following:

- As the *Source Element Size*, enter the characteristic size of the finite element.
- Specify *Edge Mapping*.
- As the material, select Air from the NX materials database. You can specify the material at the mesh creation step or later by

assigning the corresponding properties to the mesh collector.

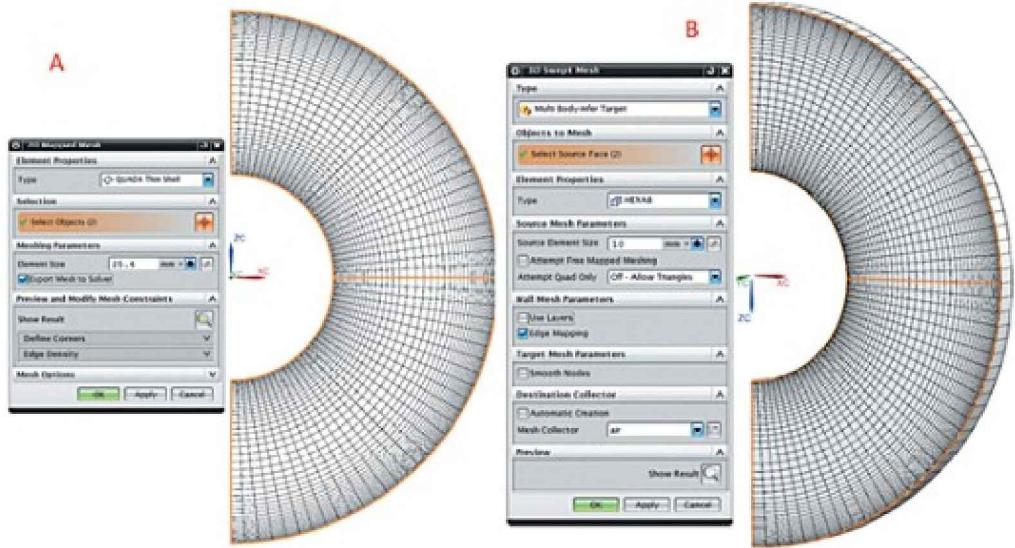


Figure 6.18. Constructing the surface mesh (A) and the 3D mesh derived from it (B)

After building the FE model, create the simulation model file (SIM). To do so, right-click the FE model in the **Simulation File View** and choose **New Simulation** in the shortcut menu. In the dialog box, specify the *NX Thermal/Flow* template, and make sure the working *Folder* and the file *Name* are specified correctly. Click **OK**.

In the **New Simulation** dialog box, keep default values and click **OK**.

In the **Solution** dialog box, specify the following: *Analysis Type – Flow*, *Solution Type – Advanced Flow*.

In the **Solution Details** tab, specify the following:

- *Turbulence Model – None – Laminar Flow*.
- Select the *Buoyancy* check box.

In the **Result Options** tab, in the *3D Flow* group, select the *Convective Flux* check box to compare simulation results with experimental data.

The direction of the gravity force is important for the solution. To make sure it is set correctly, select the *Ambient Conditions* tab and use the *Specify Vector* option in the *Gravity* group to visualize the direction. If necessary,

you can change the direction of the vector.

Then specify boundary conditions. In this example, you need to specify fixed temperature conditions at duct walls and symmetry conditions at end faces of the simulation region.

To specify temperature boundary conditions, use the **Temperature** command of the **Advanced Simulation** toolbar. In the **Temperature** dialog box, specify a temperature value of 373 K for the inside diameter and 327 K for the outside diameter (Figure 6.19).

To specify symmetry, create a **Symmetry Plane** simulation object on the **Advanced Simulation** toolbar.

Create three symmetry planes as shown in Figure 6.20.



Figure 6.19. Specifying temperature at the inside and outside diameters

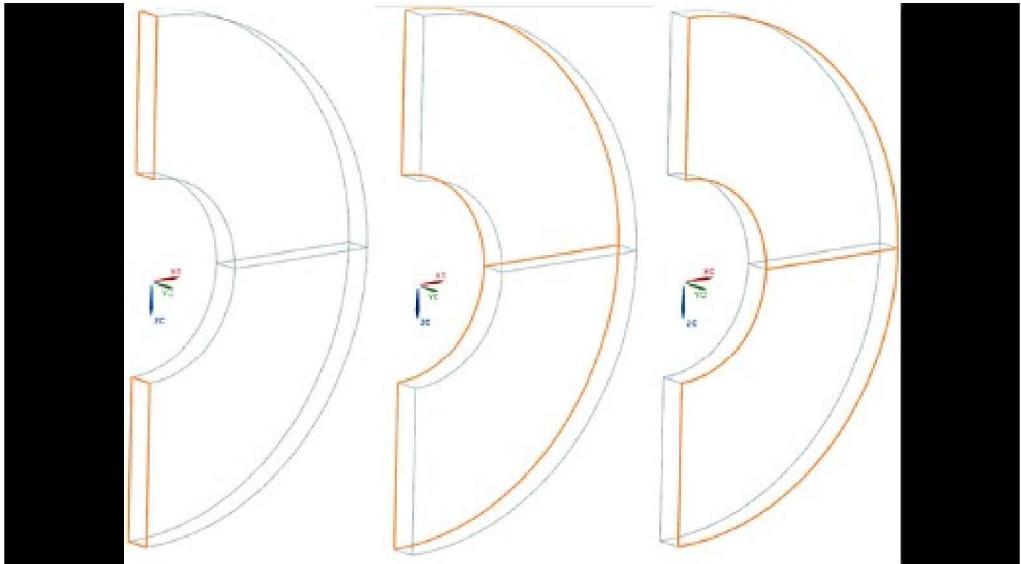
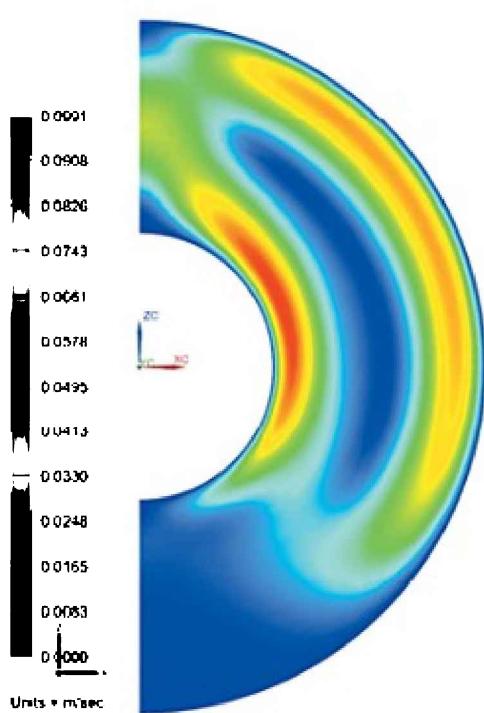


Figure 6.20. Specifying symmetry conditions

After specifying boundary conditions, you can run the simulation. When the simulation finishes, visualize the results by loading the results into the **Post Processing Navigator**. Then expand the node with the name of the solution. This node contains the list of data you can visualize. Figure 6.21 shows some of the simulation results. For a detailed description of result visualization tools, refer to Chapter 5 of Part 1.

Velocity - Element-Model



Fluid Temperature - Element-Nodal

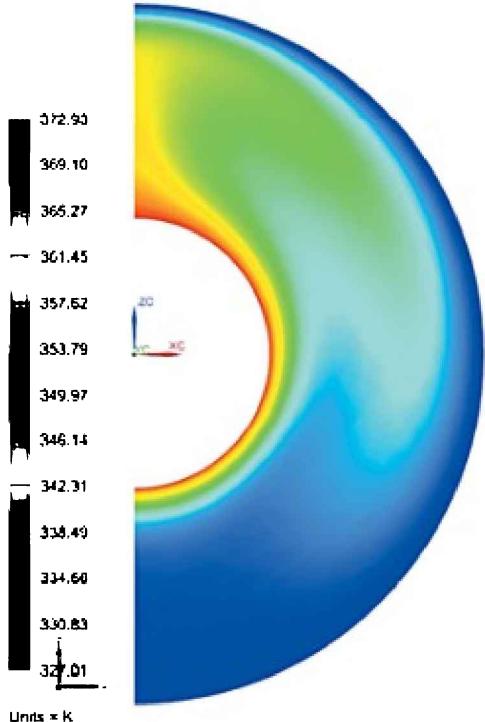


Figure 6.21. Velocity and temperature distribution for free-convection flow in the gap

Now you can compare experimental data [17, 18] to NX Advanced Flow simulation data. Figure 6.22 shows the distribution of the heat flux escaping the inner ring (A) and the distribution of temperature along the top boundary of the symmetry plane (B).

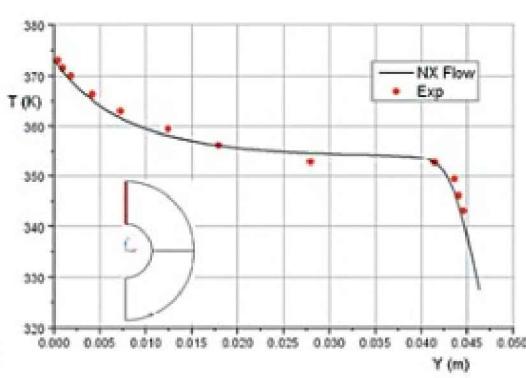
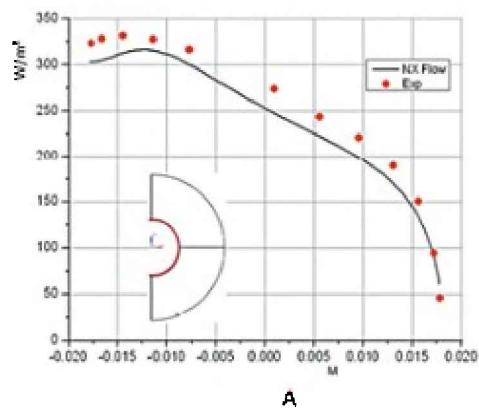


Figure 6.22. Simulation data compared with experimental data: A. Heat flux from the inner ring. B. Temperature distribution along the top boundary of

the symmetry plane

6.9.2. Mixing cold and hot flows in a cylindrical T-duct

This problem deals with flow in a T-duct. Cold fluid is injected from one end, and hot fluid is injected from the other end. The objective is obtaining data on the mixing of flows.

A sketch of the simulation region is shown in Figure 6.23. The FE mesh is constructed using the automatic *Fluid Domain* mesh generator.

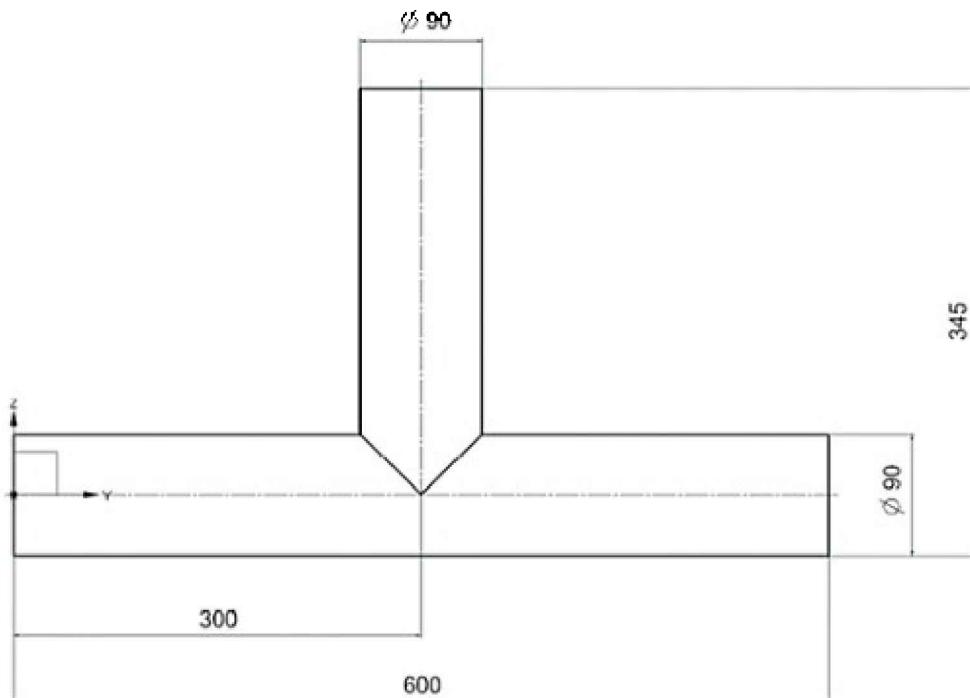


Figure 6.23. Sketch of the simulation region

To simplify subsequent mesh construction, the geometry model is split into parts using the **Split Body** command of the **Model Preparation** toolbar as shown in Figure 6.24, A. As a result, the model is represented as four parts: three cylindrical parts and a central T-shaped part. This operation is necessary to achieve a high quality surface mesh that is then used to construct the boundary layer mesh.

After preparing the geometry, create the *FEM* file.

To construct an unbroken surface mesh, use the **Mesh Mating Condition**

command, then construct the surface 2D mesh (for all surfaces except end faces) as shown in Figure 6.24, B. The resulting FE model is shown in Figure 6.25, C.

If you use the automatic mesh generator, the fluid properties must be specified in the *SIM* file of the simulation model. To create this file, in the **Simulation File View** dialog box, right-click the *FEM* file and choose **New Simulation** in the shortcut menu. In the template selection dialog box, select the *NX Thermal/Flow* template.

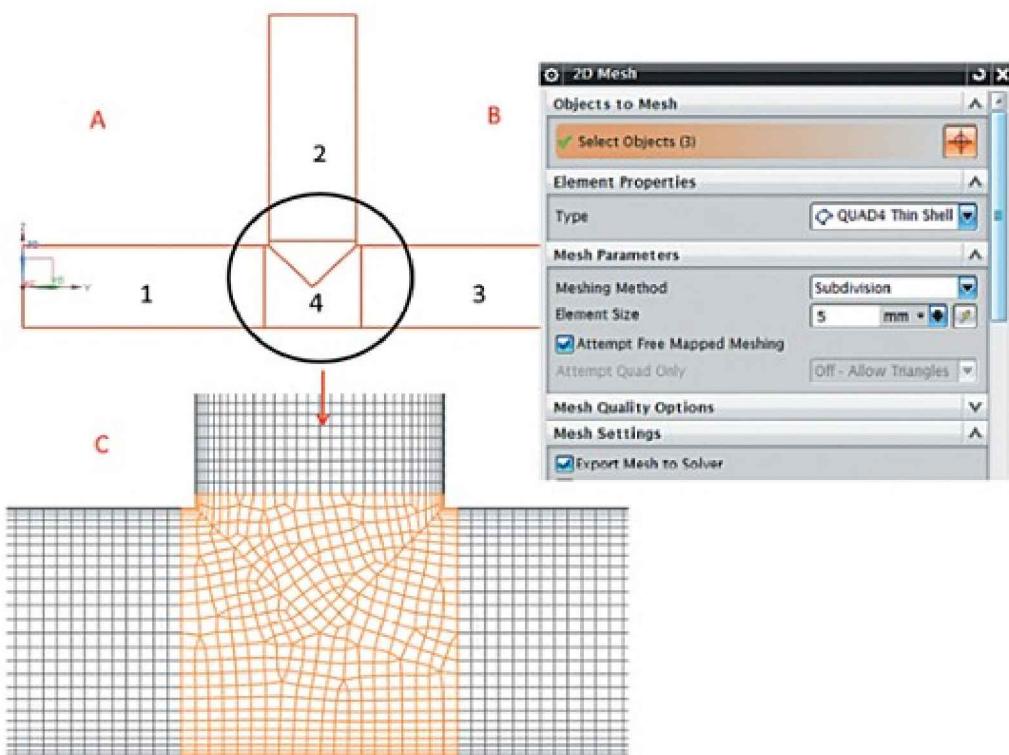


Figure 6.24. A. Preliminary splitting of the model. B. Mesh splitting parameters. C. 2D mesh

To create a volume mesh automatically, use the **Fluid Domain** command on the **Advanced Simulation** toolbar. In the **Fluid Domain** dialog box, specify the following:

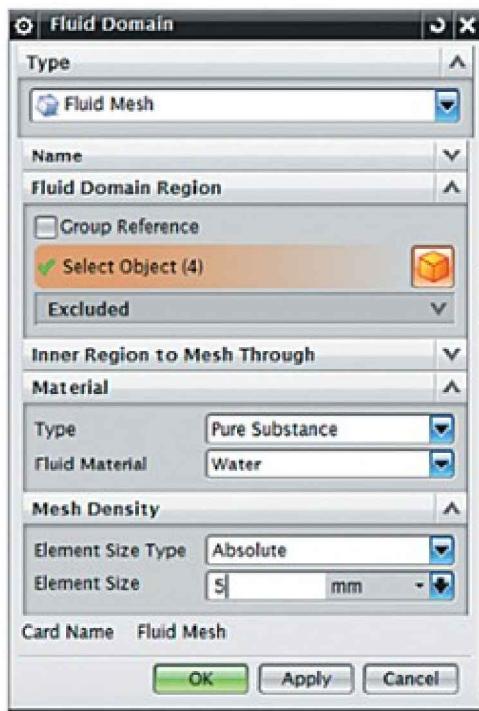
- As the **Type**, select *Fluid Mesh*.
- In the **Fluid Domain Region** group, select the entire geometry (4 objects).
- As the **Fluid Material**, specify *Water* (Figure 6.25, A).

- As the *Element Size Type*, select *Absolute*.
- As the *Element Size*, select 5 mm.

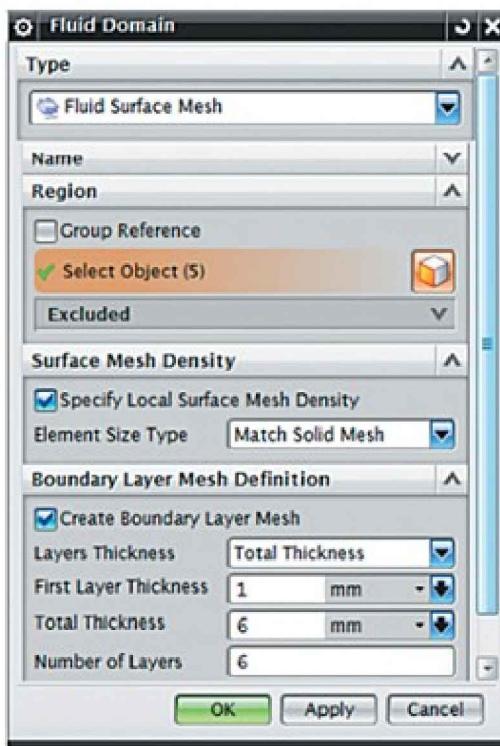
Now use the auxiliary (template) 2D mesh to construct the wall layer mesh. To do so, in the **Fluid Domain** dialog box, specify the following (Figure 6.25, B):

- As the *Type*, select *Fluid Surface Mesh*.
- Select the *Specify Local Surface Mesh Density* check box to make the option of using the existing 2D mesh available.
- As the *Element Size Type*, select *Match Solid Mesh*.
- Select the *Create Boundary Layer Mesh* check box.
- As the *Layers Thickness*, select *Total Thickness*.
- As the *First Layer Thickness*, enter 1 mm.
- As the *Total Thickness*, enter 6 mm.
- As the *Number of Layers*, type 6.

Click **OK**.



A



B

Figure 6.25. FE mesh parameters

Specify velocity and temperature as boundary conditions at the inlet boundaries and free outflow condition at the outlet boundary. Locations of boundaries and inlet flow parameters are shown in Figure 6.26. For example, at the left-hand boundary, the velocity of the flow is 0.7 m/s at 25 °C. At the top boundary, the injection velocity is 0.5 m/s, and the temperature of the fluid is 90 °C. At the right-hand boundary, free outflow condition is applied.

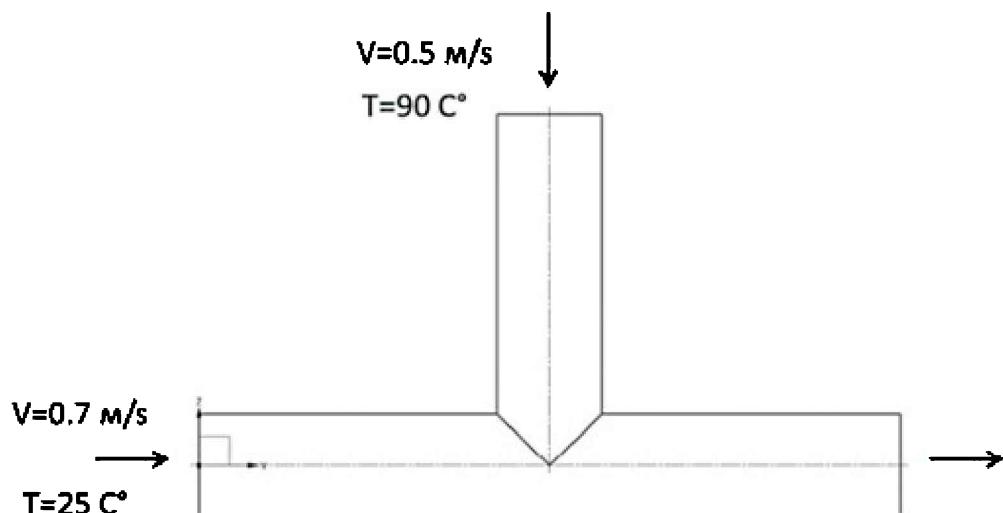


Figure 6.26. Boundary conditions

You can specify boundary conditions with the **Flow Boundary Condition** command on the **Advanced Simulation** toolbar. To set up the velocity and temperature of the flow, in the **Flow Boundary Condition** dialog box, specify the following (Figure 6.27):

- Type –*Inlet Flow*.
- Mode – *Velocity*.
- Click **Create Modeling Object** for the *External Conditions* parameter and in the **External Conditions** dialog box, specify the inlet temperature value of 25 °C.

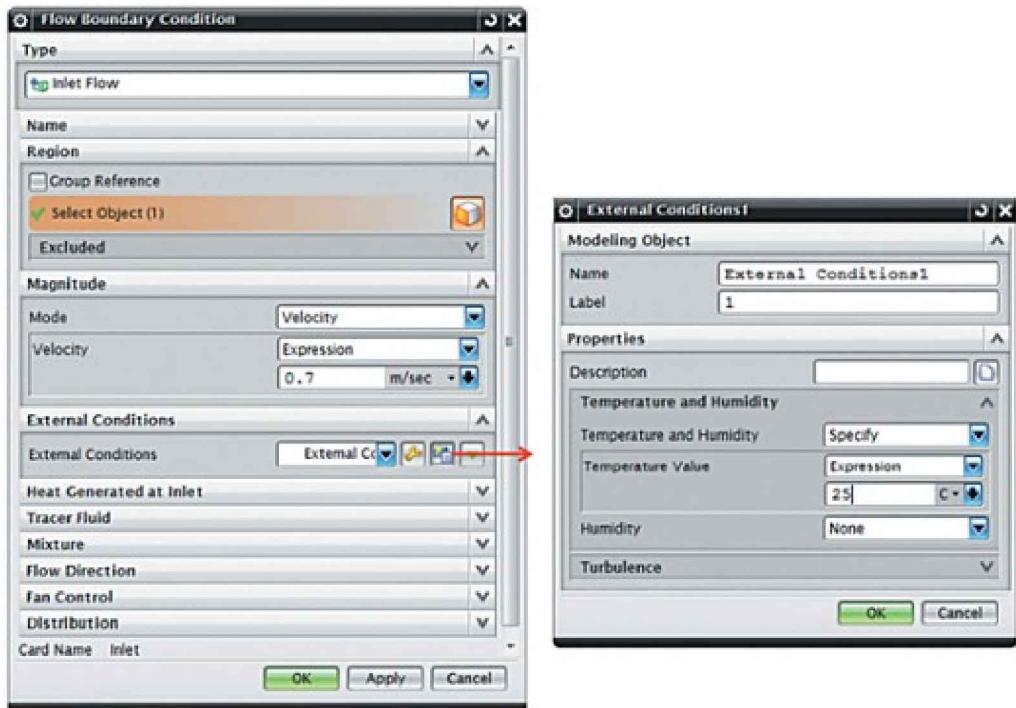


Figure 6.27. Specifying inlet conditions for the simulation region

In accordance with the problem formulation, specify a temperature value of 90 °C for the second inlet boundary.

Outlet conditions are also specified in the **Flow Boundary Condition** dialog box. In the *Type* group, select *Opening*. To do so, select the relevant boundary (Figure 6.26) and click **OK**.

Use the *K-Epsilon* model as the turbulence model. To do so, in the **Solution** dialog box select the *Solution Details* tab and in the *Turbulence Model* list, select *K-Epsilon*.

The problem is now ready to simulate. It is advisable to visualize the automatically generated mesh before running the simulation. To do so, run the simulation using the *Write Solver Input File* option (Figure 6.28). The system then starts building the mesh and if that process finishes successfully, a *Results* node is added to the solution tree. Double-click the solution result to enter visualization mode. At this stage, only mesh model information is available for visualization because the simulation is not yet complete (Figure 6.29).

If the quality of the mesh is satisfactory, run the complete simulation. To do so, switch the mode from *Write Solver Input File* to *Solve* (Figure 6.28).



Figure 6.28. Generating and saving the mesh if automatic generator is used

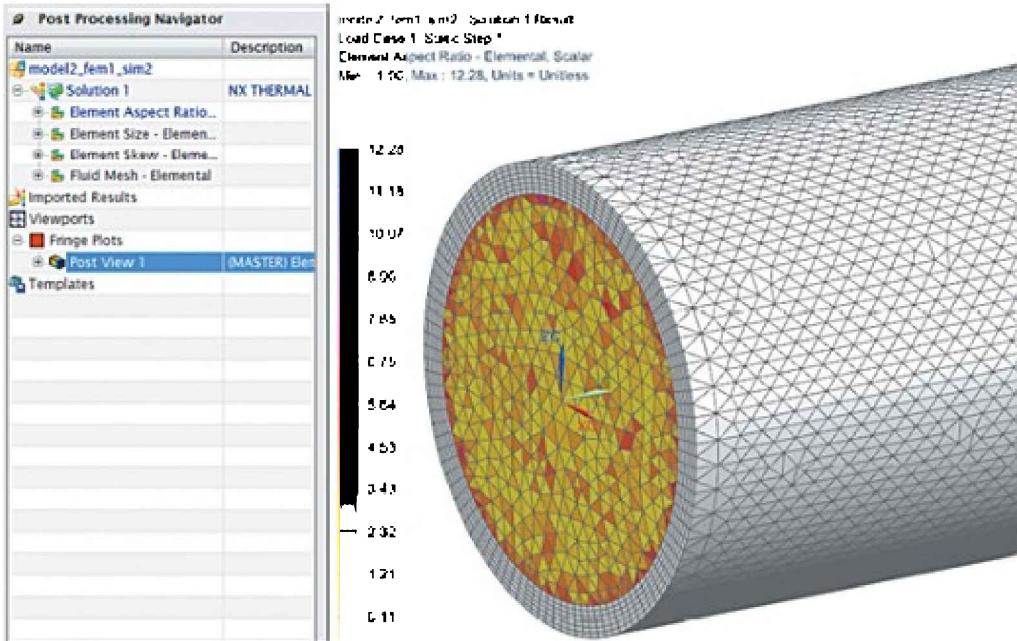


Figure 6.29. Visualizing the mesh that was constructed using Fluid Domain

The mesh can also be constructed in the classical way without the **Fluid Domain** command. You can generate a mesh that contains only hexahedral elements for this geometry. To do so, carry out additional preparation of

geometry (Figure 6.30, A) and specify the splitting mode for edges (Figure 6.30, B). The resulting mesh is shown in Figure 6.30, C.

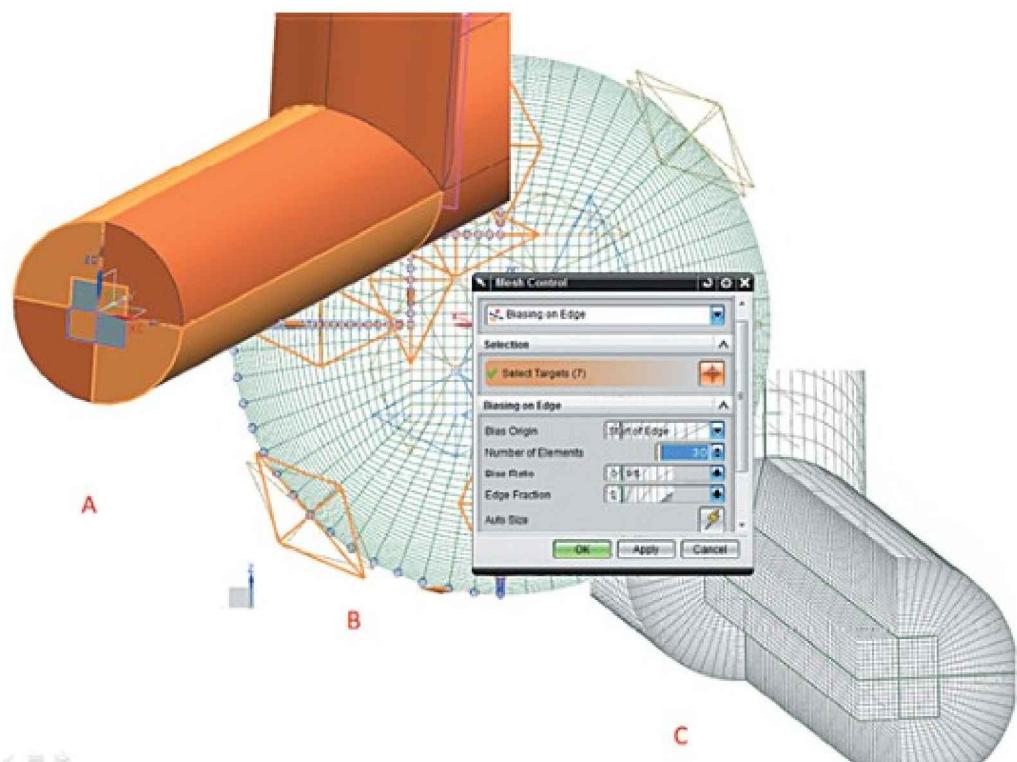


Figure 6.30. Constructing a hex mesh. A. Preparing the geometry. B. Specifying edge splitting. C. A segment of the mesh

After running the simulation, the **Solution Monitor** dialog box appears. It traces the solution progress by displaying text information (Figure 6.31, A) and convergence graphs of primary parameters (Figure 6.31, B).

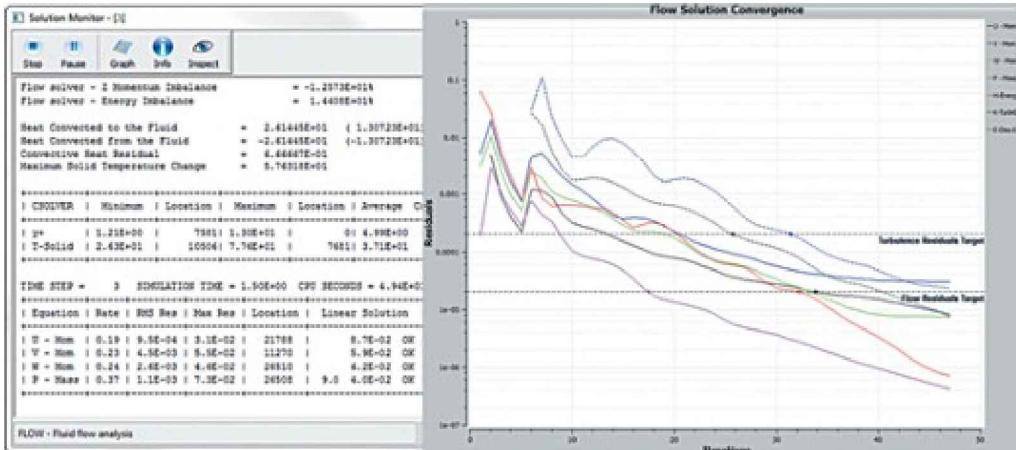


Figure 6.31. A. Solution progress information. B. Convergence graphs

When the simulation finishes, load the results into the **Post Processing Navigator**. Figure 6.32 shows temperature distributions calculated for automatic (A) and hexahedral (B) meshes.

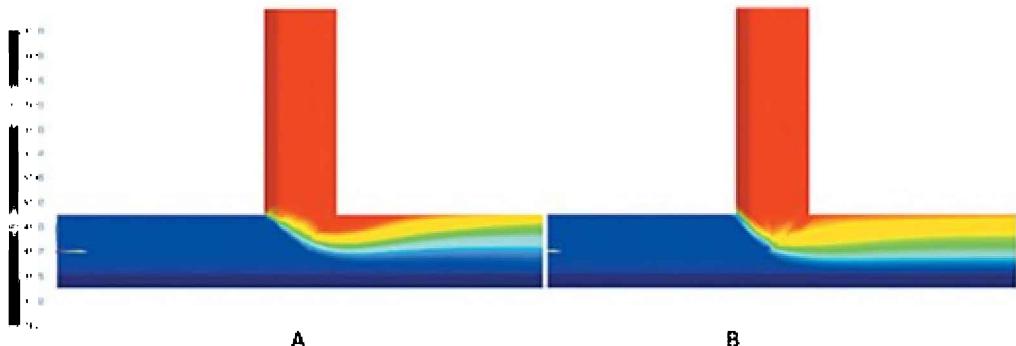


Figure 6.32. Temperature distribution for solutions results with automatic (A) and hexahedral (B) meshes

References

1. Goncharov, P. S., Yeltsov, M. Yu., Korshikov, S. B., Laptev, I. V., Osiyuk, V. A., NX for the Mechanical Design Engineer, DMK Press, Moscow, 2010. – 501 p.
2. Zenkevich, O., Finite Element Method in Engineering, Mir, Moscow, 1975. – 542 p.
3. Strength, Oscillatory Stability: Reference Guide in 3 vol. Vol. 3 / edited by Birger, I. A. and Panovko, Ya. G., Mashinostroenie, Moscow, 1968. – 567 p.
4. NX Nastran User's Guide. – 2011.
5. NX Nastran Numerical Methods User's Guide. – 2011.
6. NX Nastran Basic Nonlinear Analysis User's Guide. – 2011.
7. NX Nastran Basic Dynamic Analysis User's Guide. – 2011.
8. NX Nastran Quick Reference Guide. – 2011.
9. Vibrations in Engineering: Reference Guide in 6 vol. Mashinostroenie, Moscow, 1978. Vol. 1: Oscillations of Linear Systems / edited by Bolotin, V. V., 1978. – 352 p.
10. NX Nastran Superelement User's Guide. – 2011.
11. NX Nastran Numerical Methods User's Guide. – 2011.
12. Holman, J. P., Heat Transfer, 7th ed., pp. 418–419, Example 8–7, McGraw-Hill, New York, 1990.
13. Oppenheim, A. K., Radiation Analysis by the Network Method, Trans. ASME, vol. 78, pp. 725–35, 1956.
14. Belov, I. A., Isaev, S. A., Modeling Turbulent Flows: workbook, Baltic State Technical University, St. Petersburg, 2001. – 108 p.
15. Yun, A. A., A. Modeling Turbulent Flows, Librokom, Moscow, 2010. – 351 p.
16. Bradshaw, P., An introduction to turbulence and its measurement, Mir, Moscow, 1974. – 278 p.
17. Kuehn, T. H., Goldstein, R. J., An Experimental Study of Natural Convection Heat Transfer in Concentric and Eccentric Horizontal Cylindrical Annuli, Journal of Heat Transfer, 100: pp. 635–640, 1978.
18. Kuehn, T. H., Goldstein, R. J., An Experimental and Theoretical Study of Natural Convection in the Annulus Between Horizontal Concentric Cylinders, Journal of Fluid Mechanics, 74: pp. 695–719, 1976.
19. O. Daniel, I. M. Ishai, Engineering Mechanics of Composite Materials.

- Oxford, 2nd ed., 2006.
20. J.-M. Berthelot, *Matériaux composites, Comportement mécanique et analyse des structures*. Masson, 2nd ed., 1996.
21. Subodh K. Mital, Pappu L. N. Murthy, Robert K. Goldberg, *Micromechanics for particulate reinforced composites*, NASA TM-107276, 1996.
22. J. N. Reddy, *Mechanics of Laminated Composite Plates: Theory and Analysis*. CRC Press, 1997.
23. NX Nastran 8 Advanced Nonlinear Theory and Modeling Guide. – 2011.
24. Malinin, N. N., *Applied theory of plasticity and creep*, Mashinostroenie, Moscow, 1968. – 400 p.
25. NX Nastran Element Library Reference. – 2011.
26. NX Nastran Design Sensitivity and Optimization User's Guide. – 2011.
27. Borovkov, A. I., Zubov A. V., *A finite element solution of steady-state and unsteady-state nonlinear heat conductivity problems of heterogeneous anisotropic media*. VINITI, N897-B94, 1994. – 44 p.