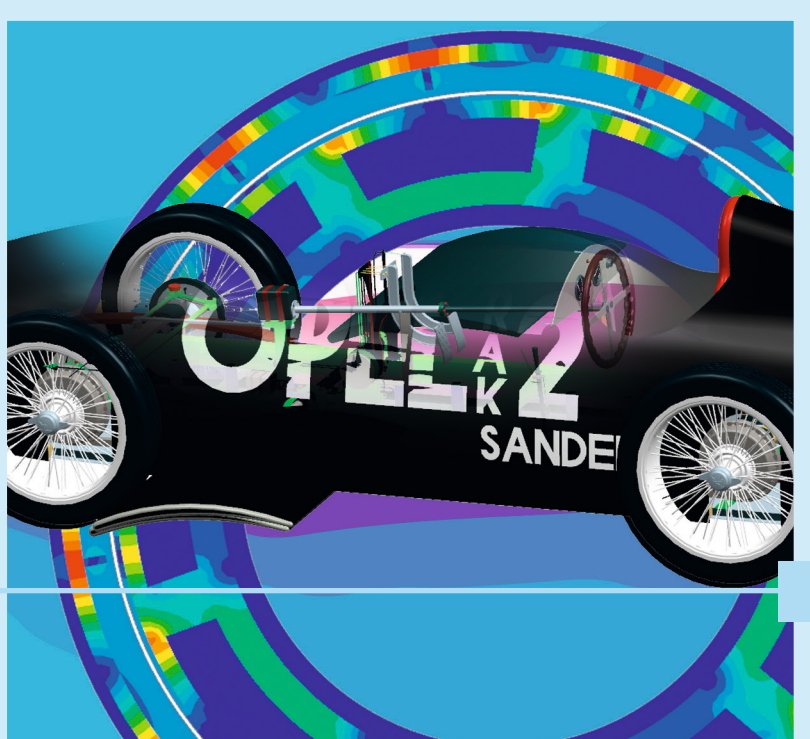


Reiner Anderl
Peter Binde

Simulations with NX / Simcenter 3D

Kinematics, FEA, CFD, EM and Data Management



2nd Edition

HANSER

Anderl/Binde

Simulations with NX / Simcenter 3D

Reiner Anderl
Peter Binde

Simulations with NX / Simcenter 3D

Kinematics, FEA, CFD, EM and Data Management

2nd Edition



HANSER

The Authors:

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

Dr.-Ing. Peter Binde, Dr. Binde Ingenieure, Design & Engineering GmbH, Wiesbaden, Germany

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

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



The use of general descriptive names, trademarks, etc., in this publication, even if the former are not especially identified, is not to be taken as a sign that such names, as understood by the Trade Marks and Merchandise Marks Act, may accordingly be used freely by anyone. While the advice and information in this book are believed to be true and accurate at the date of going to press, neither the authors nor the editors nor the publisher can accept any legal responsibility for any errors or omissions that may be made. The publisher makes no warranty, express or implied, with respect to the material contained herein. The final determination of the suitability of any information for the use contemplated for a given application remains the sole responsibility of the user.

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

Bibliografische Information der deutschen Bibliothek:

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

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

ISBN 978-1-56990-712-2

E-Book ISBN 978-1-56990-713-9

© Carl Hanser Verlag, Munich 2018

Editorial Management: Julia Stepp

Production Management: Isabell Eschenberg, Christin Jahn

Cover Concept: Marc Müller-Bremer, www.rebranding.de, München, Germany

Cover Design: Stephan Rönigk

Typesetted by Kösel Media GmbH, Krugzell, Germany

Printed and bound by Kösel GmbH & Co. KG, Krugzell, Germany

Printed in Germany

Contents

Preface	IX
Preamble	XI
1 Introduction	1
1.1 Learning Tasks, Learning Objectives, and Important Prerequisites for Working with the Book	4
1.2 Work Environments	6
1.3 Working with the Book	7
2 Motion (Multibody Dynamics)	11
2.1 Introduction and Theory	11
2.1.1 Simulation Methods	12
2.1.2 Restrictions	14
2.1.3 Classifications of MBD	14
2.2 Learning Tasks for Kinematics	16
2.2.1 Steering Gear	16
2.2.2 Top-down Development of the Steering Lever Kinematics	34
2.2.3 Collision Check on Overall Model of the Steering System	52
2.3 Learning Tasks on Dynamics	62
2.3.1 Drop Test on Vehicle Wheel	62
2.4 Learning Tasks on Co-Simulation	70
2.4.1 Balancing a Pendulum	71

3	Design Simulation FEM (Nastran)	81
3.1	Introduction and Theory	82
3.1.1	Linear Statics	83
3.1.2	Non-linear Effects	85
3.1.3	Influence of the Mesh Fineness	87
3.1.4	Singularities	88
3.1.5	Eigenfrequencies	89
3.1.6	Heat Transfer	91
3.1.7	Linear Buckling	92
3.2	Learning Tasks on Design Simulations	92
3.2.1	Notch Stress at the Steering Lever (Sol101)	93
3.2.2	Temperature Field in a Rocket (Sol153)	140
4	Structural (FEM)	149
4.1	Introduction	150
4.1.1	Sol 101: Linear Statics and Contact	151
4.1.2	Sol 103: Eigenfrequencies	152
4.1.3	Sol 106: Non-linear Statics	152
4.1.4	Sol 601/701: Advanced Non-linear	152
4.2	Learning tasks on Linear Analysis and Contact (Sol 101/103)	155
4.2.1	Stiffness of the Vehicle Frame	155
4.2.2	Size and Calculation of a Coil Spring	186
4.2.3	Eigenfrequencies of the Vehicle Frame	200
4.2.4	Clamping Seat Analysis on the Wing Lever with Contact	207
4.3	Learning Tasks Basic Non-Linear Analysis (Sol 106)	230
4.3.1	Analysis of the Leaf Spring with Large Deformation	230
4.3.2	Plastic Deformation of the Brake Pedal	240
4.4	Learning Tasks Advanced Non-linear (Sol 601)	249
4.4.1	Snap Hook with Contact and Large Deformation	249
5	Thermal/Flow (CFD)	271
5.1	Principle of Numerical Flow Analysis	272
5.2	Learning Tasks (Simcenter Flow)	273
5.2.1	Flow Behavior and Lift Forces at a Wing Profile	273
6	Electromagnetics (EM)	297
6.1	Principles of Electromagnetic Analysis	298
6.1.1	Electromagnetic Models	299
6.1.2	Maxwell Equations	300

6.1.3	Material Equations	302
6.1.4	Model Selection	304
6.1.5	Electrostatics	306
6.1.6	Electrokinetics	306
6.1.7	Electrodynamics	307
6.1.8	Magnetostatics	307
6.1.9	Magnetodynamics	307
6.1.10	Full Wave (High Frequency)	308
6.2	Weak Formulation of EM Problems	308
6.3	Installation and Licensing	309
6.4	Learning Tasks (EM)	312
6.4.1	Coil with Core, Axisymmetric	312
6.4.2	Coil with Core, 3D	327
6.4.3	Electric Motor	331
7	Management of Analysis and Simulation Data	353
7.1	Introduction and Theory	353
7.1.1	CAD/CAE Integration Issues	353
7.1.2	Solutions with Teamcenter for Simulation	354
7.2	Learning Tasks on Teamcenter for Simulation	356
7.2.1	Carrying out an Simcenter 3D Structural Analysis in Teamcenter	357
7.2.2	Which CAD Model Belongs to which FEM Model?	365
7.2.3	Creating Revisions	367
8	Manual Analysis of a FEM Example	373
8.1	Task Formulation	373
8.2	Idealization and Choice of a Theory	374
8.3	Analytical Solution	374
8.4	Space Discretization for FEM	375
8.5	Setting up and Solving the FEA System of Equations	376
8.6	Analytical Solution Compared with Solution from FEA	378
	Bibliography	381
	Index	385

Preface

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

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

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

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

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

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

June 2018

Prof. Dr.-Ing. Reiner Anderl

Dr.-Ing. Peter Binde

Preamble

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

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

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

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

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

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

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

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

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

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

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

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

Eckardt Niederauer

Portfolio Manager Simcenter 3D

Siemens PLM Software

www.siemens.com/plm/simcenter

1

Introduction

Engineering science has been faced with significant changes during the past two decades. These changes have been driven by a powerful development of information and communication technologies and their introduction into both the product development process and the products themselves. In essence, it has enabled computer integrated virtual product development, based on integrated 3D modeling, analysis, simulation, and optimization.

The primary goal of virtual product development is the efficient development of innovative product solutions that satisfy customer needs. Consequently, the integration of computer-based methods into the digital workflow of the product development process has become critical to the success of virtual product development.

Engineering, designing, and detailing are essential tasks for the development of innovative product solutions, as well as the ability to accurately predict the product's behavior subject to the multitude of potential use cases and operating conditions. Fortunately, with the continuous improvement of information and communication technologies, and with the subsequent improvements in integration of computer aided design, analysis, simulation, and optimization, it has become increasingly easier to successfully perform these essential product development tasks.

Information and communication technologies (ICT) are increasingly influencing the product development process, in particular as the product development process becomes increasingly virtualized. This influence results from:

- rapid information acquisition from sources worldwide
- the availability of new computer-based methods for product development and design – such as for product modeling, e.g., parametric, feature based, and knowledge-driven CAD; analysis, simulation and optimization, e.g., finite element analysis (FEA), multi-body simulation (MBS), and computational fluid dynamics (CFD); rapid validation and verification, e.g., digital mock-up (DMU); virtual and physical prototyping, e.g., virtually by using virtual and augmented reality, or physically by using generative manufacturing machines; and processing product data in successive process chains (so-called CAX processes, e.g. for advanced manufacturing including additive manufacturing)
- mapping of the organizational and workflow structures into product data management (PDM) systems, with the aim to provide easy, intuitive, and immediate access to development status, progress, and results

Impact of information and communication technologies on product development

The concept of virtual product development has clearly been shaped by the deep penetration of ICT into the product development process, to provide seamless flows of product data. Virtual product development can be systematically achieved over an escalating set of levels (see Figure 1.1). These levels consist of

- 3D CAD;
- digital mock-ups;
- virtual prototyping;
- virtual product analysis and simulation; and
- virtual factory.

In addition, analysis and simulation are becoming increasingly important due to Industry 4.0. The Industry 4.0 initiative stands for the 4th industrial revolution and moves organization and control of the entire added value chain to the next efficiency level across all phases of the product lifecycle. The focus lies on increasing the flexibility of production and on satisfying individual customer demands. A prerequisite is the availability of information in real time in order to optimally control the added-value flow. This requires interconnected and communication-capable cyber-physical systems. This approach also requires so-called digital twins, which are a digital representation of physically existing components and products. Thus, digital twins offer fundamental approaches for performing realistic analysis and simulations.

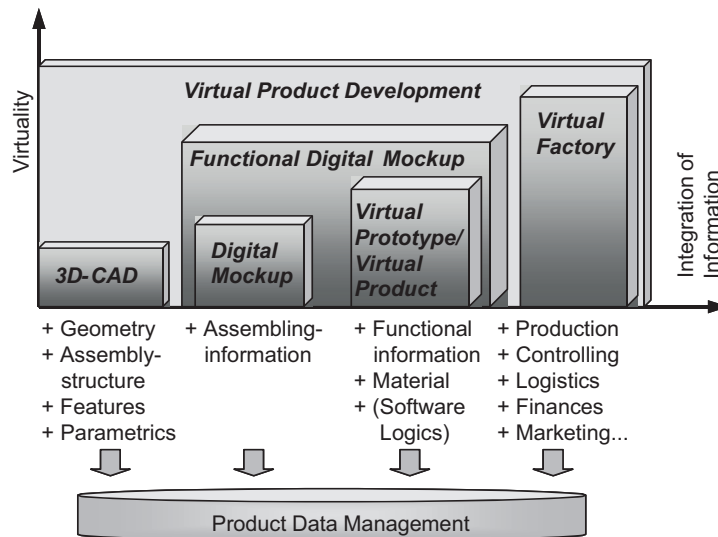


Figure 1.1 Levels of virtual product development

3D CAD is the fundamental basis.

3D CAD is the fundamental basis for describing product geometry, geometrically represented as solid geometry. This digital product representation involves single-part modeling as well as assembly modeling, and typically describes a product structure. This modeling approach is mainly based on features and parametric modelling.

Digital mock-ups (DMU) provide a digital representation of the product structure as well as part and assembly geometry. The geometry representation is typically approximated using tessellation and triangulation approaches. As the part and assembly models are represented as solid geometry and complemented by material data, product properties, such as mass and center of gravity, can be derived. Digital mock-ups enable virtual prototyping for simulating assembly and disassembly processes and for investigating collision detection.

DMU

Virtual prototypes – often referred to as digital prototypes – include material and physical properties in addition to part geometries and product structures. Therefore, these prototypes therefore can be used to simulate the functional and physical behavior of a product and to visualize its behavior. The functional and physical modeling within a virtual prototype tends to be application specific and discipline specific. Typical applications include stress analysis using finite element analysis (FEA) based on the finite element method (FEM), multi-body simulation/dynamics (MBS/MBD), or fluid dynamic simulation using computational fluid dynamics (CFD). Simulations may also integrate thermal analysis, electromagnetic analysis (EM), or kinematic analysis, or their combinations, typically based on FEA, to more fully investigate and understand the product behavior.

The most important simulation methods are FEA, MBS and CFD.

The term “virtual product” refers to the aggregation of a product’s physical properties together with its logical dependencies to produce a comprehensive, interoperable product model.

The term “virtual factory” refers to the digital representation of a factory, including its physical properties and manufacturing processes. The objective is to facilitate analysis, simulation, and optimization of factory operations, including material flows, logistics, and order processing.

While the terms “digital prototype” and “virtual product” refer to digital representations of non-existent products and to the development of optimized product solutions, the term “digital twin” refers to digital representations of physically existing, manufactured products with their current physical dimensions and properties.

Product data resulting from the application of the various modeling, analysis, simulation, and optimization software systems is stored as files in the product data management (PDM) system, enriched by meta-data representing organizational and workflow information such as release status, effectivity, identification, classification, and version numbers.

The PDM system manages all product data generated through virtual product development.

The increasing use of 3D CAD in industry leads to an increasing need to integrate numerical analysis, simulation, and optimization methods and tools. With this integration, product data, once described or generated, can then be used and reused in successive processes to avoid manual redescription and errors, as well as to identify errors early. This practice enhances product quality and increases the efficiency of virtual product development process and successive physical product realization.

■ 1.1 Learning Tasks, Learning Objectives, and Important Prerequisites for Working with the Book

The training content is taught by using methodic examples.

Based on the objective to use 3D CAD data for analysis, simulation and optimization, the question of how 3D CAD data is to be used further, follows. For this purpose, representative use case scenarios for the procedures of finite element analysis, multi-body simulation, fluid dynamics and electromagnetic simulation have been developed in this book, by which the integration of modeling, analysis, and simulation will be presented. The outlined use case scenarios are based on Simcenter 3D 12 and the 3D CAD system NX with its integral analysis and simulation modules.

To facilitate the understanding of the methodology and to increase the training efficiency, a single contiguous assembly was chosen for most learning tasks of this book. It is the 3D CAD model of the legendary Opel RAK2 (Figure 1.2) which was created in student projects as a 3D CAD solid model at the division Computer Integrated Design (DiK) at TU Darmstadt.

In 1928, Fritz von Opel, grandson of Adam Opel, built the rocket driven cars RAK1 and RAK2 for testing purposes. With RAK2 he reached a speed record of 238 km/h on Berlin high speed track AVUS on 23rd of May in 1928. The RAK2 was powered by 24 solid-fuel rockets, which were filled with 120 kg of fuel. This attempt to establish the rocket engine was followed by further attempts by road, rail and air.

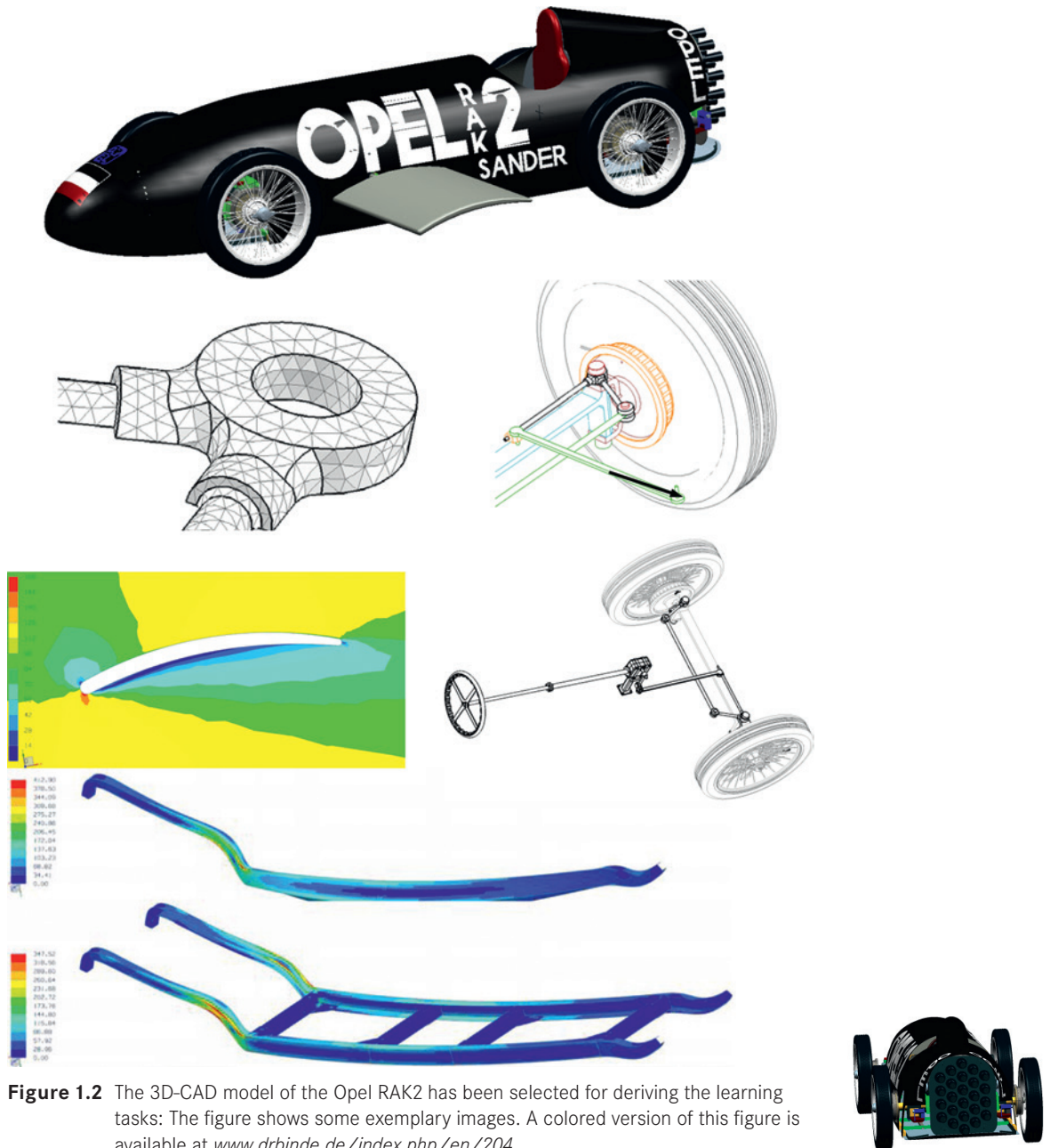


Figure 1.2 The 3D-CAD model of the Opel RAK2 has been selected for deriving the learning tasks: The figure shows some exemplary images. A colored version of this figure is available at www.drbinde.de/index.php/en/204.

Prerequisites for working with the book

All 3D-CAD and analysis data used or created in the learning tasks are stored in an archive file for download (see following web link) and should be used to reproduce the examples.



The archive file can be downloaded at www.drbinde.de/index.php/en/204. This file contains all 3D-CAD models, analysis and result files. Furthermore, the installation files for the electromagnetic solver are included in this file.

Objective is to build a fundamental knowledge about engineering, design, analysis and simulation.

The training content is taught by using practical examples. Functions of the Simcenter 3D system are therefore not explained stand-alone, but always in the context of an example. Since its similarity to learning from physical-world projects, this method is efficient, memorable and adheres to modern didactic.

The structure of the chapters is based on the didactic concept of continuous learning progress, but it is also built on the fundamentals of working with 3D CAD, in particular the Simcenter 3D system. Therefore, knowledge about the description of 3D parametric models for single parts and assemblies is required.

The learning objective is to convey knowledge to designers or analysis engineers that she or he needs to solve appropriate tasks using finite element analysis, multibody simulation, and flow simulation supported by Simcenter 3D, and to develop an understanding of these technologies. However, solving complex practical problems needs more engineering experience and knowledge and probably exceeds the examples presented in this book. A novice, however, is able to develop into an expert by working through theory and as many practical tasks as possible. He thereby collects valuable experience and understands the engineering background. The gained experience thus results from successfully developed projects. This book, with its learning examples, provides important basic experiences and thus forms the basis for a systematic expandable knowledge enriched by experience.

■ 1.2 Work Environments

Engineering simulation problems can be subdivided into four classes: rigid bodies, elastic bodies, fluids, and electrical/magnetic bodies (Figure 1.3). Rigid body systems are simulated using Multibody Dynamics programs (MBD); elastic and also electric/magnetic bodies are simulated using the Finite Element Method (FEM); and flow tasks are simulated using Computational Fluid Dynamics (CFD).

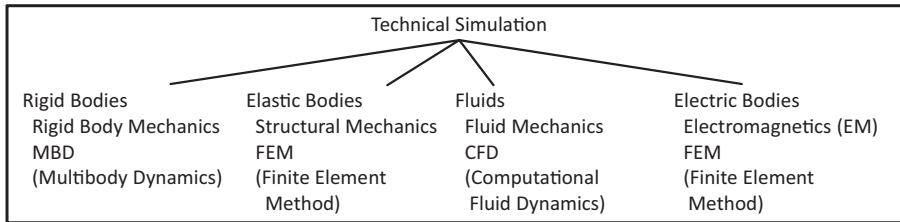


Figure 1.3 Technical simulation can be roughly divided into four parts.

Within the Simcenter 3D system, there are several modules for engineering simulation. The three most important ones used in this book are (in addition to some others not covered here):

- *Motion Simulation* for kinematic and dynamic motion simulations with MBD
- *Design-Simulation FEM* for simple structural, thermal, and eigenfrequency analysis
- *Simcenter Pre/Post* for complex simulation tasks: This module is intended for engineers that focus on analysis. Additional simulation capabilities include modeling and simulation of complex assembly structures and the choice of various solvers for addressing particular physical phenomena. The problem domains that can be addressed include structural mechanics, thermodynamics, fluid mechanics, and electromagnetism (EM).

The working environments for these modules have a common interface, and default to only include those features that are useful in the selected context.

This book reflects in detail these working environments. Possibilities and limitations will be illustrated by examples.

The Simcenter 3D system provides many modules for technical simulation.

■ 1.3 Working with the Book

The book contains chapters on the following topics:

- Motion Simulation (MBD)
- Design Simulation (FEM)
- Structural (FEM)
- Thermal/Flow (CFD)
- Electromagnetics (EM)
- Management of analysis and simulation data (Teamcenter for Simulation)
- Manual analysis of an FEM example

First, we will explore motion simulation (Chapter 2) because this class of analysis is common in engineering design and is usually carried out first. The joint forces that are determined here are often used in subsequent strength analysis using FEM.

Structure of the book

The joint forces are calculated in Motion Simulation.

The chapters can be largely worked through independently. That means, those who do not care for motion simulation, can skip this chapter. One exception: Those interested in FEM and “Structural (FEM)” (Chapter 4) should first read the chapter on “Design Simulation FEM (Nastran)” (Chapter 3) to attain the necessary prerequisites.

-The hurried reader can directly start with the examples.

At the beginning of each chapter an introduction to the principles of each topic is given. For the analysis newcomer these statements might sound very theoretical and difficult. But this should not discourage to begin with the learning tasks on this subject, which are in focus. Explanations in the learning tasks typically build on the principles of the introductions. They are a clarification and expansion of them. A hurried reader can therefore skip the introductions, and might go straight to the learning tasks.



The download files belonging to this book (www.drbinde.de/index.php/en/204) contain the *RAK2* folder. This folder includes all outlined learning tasks for the areas motion, structural, thermal, and flow simulation. A second folder named *EM* contains installation files and examples for electro-magnetics. The download file also contains solution files so that any result can be looked up in it. For working through the book this entire file should be unpacked and copied to a directory on the hard disk of the computer.

In each case, the first example is of fundamental nature.

The learning tasks of a chapter can best be worked through in the order given, because all learning content builds on each other. In Motion and Design Simulation, as well as EM, the first learning task is a basic example. All important principles and foundations are taught here, which are necessary to understand and build the following learning tasks.

Pin icons indicate steps to be performed.

When describing the learning tasks, there is a distinction between background explanations and steps to be carried out (mouse clicks in NX/Simcenter 3D). Steps to be carried out are always marked with the pin icon:

 Here a step is described that needs to be carried out.

Very hurried readers can therefore skip the background explanations (hopefully, they understand intuitively quite a bit) and jump straight from one pin icon to the next.

NX/Simcenter 3D installation and computer performance

To work through the learning tasks, a computer with NX or Simcenter 3D installation must be available. The examples were calculated by NX/Simcenter 3D 12, but should also work in other versions. With a normal installation of NX/Simcenter 3D 12, all required modules for simulation, especially the NX Nastran solver, are automatically installed. It is, other than with previous NX/Simcenter 3D versions, no longer required to define specific environment variables for the simulation manually.

Only for electromagnetic simulation (Chapter 6), the installation of some additional files is required. This is explained at the beginning of the chapter.

In addition, the computer hardware should preferably be well equipped. We would like to give the following recommendations:

- Processor: The highest possible clock frequency is essential for all simulation problems.
- Multi-Processor: For FEM analysis and some thermal analysis, the use of multiple processors is supported.

- **Memory:** FEA, thermal and fluid flow analysis need quite some memory. There is a simple rule: the more, the better. To work through the examples in this book, we recommend at least 8 GB of main memory.
- **Hard drive:** Again, there should be enough free disk space available. For the examples in this book we recommend at least 4 GB.
- **32-/64-bit operating system:** For medium to large analysis models 64-bit architecture must be selected, since much more memory can be addressed here. Nowadays the Simcenter 3D installation will only run on 64-bit systems.

For more information on these topics, we recommend reading the documents for parallel-processing [nxn_parallel] and for efficient memory usage with NX Nastran [nxn_num].

For motion analysis, there are three solvers available: Adams, RecurDyn and Simcenter Motion. The learning tasks in this book were carried out with Simcenter Motion solver, but also run with Adams or RecurDyn.

Presetting the motion solver

Well, our introduction is now complete. We wish you fun and learning success!

Bibliography

- [nxn_num] NX Nastran Numerical Methods User's Guide. Online documentation to NX Nastran
- [nxn_parallel] NX Nastran Parallel Processing User's Guide. Online documentation to NX Nastran

2

Motion (Multibody Dynamics)

In Section 2.1, first the theory, limitations, special effects and rules of this discipline are represented. This is followed by kinematic learning tasks, which start with a basic example (Section 2.2.1). In the second learning task, principle sketches and kinematics are used to support the early design phase (Section 2.2.2). In the third task, collisions and assembling of various subkinematics are treated (Section 2.2.3). The fourth example deals with dynamic problems and the simulation of contact (Section 2.3.1) and the final task deals with the coupling of Simcenter Motion with MATLAB® Simulink® for the so-called co-simulation (Section 2.4.1).

Content of the chapter

■ 2.1 Introduction and Theory

Simcenter Motion offers the ability to control the movements of a design which is otherwise a statically developed machine. This allows a better understanding of the mechanism and it can be checked if the movement of the components leads to any collisions. It also can be checked if the machine can carry out the desired movement, or even reach certain positions. Often, one of the tasks of the simulation is to adjust the geometrical dimensions in a suitable way. The use of parametric CAD models is an important way to create variations.

Use cases and benefits of Simcenter motion in practice

But also and especially in the early stage of the design process when only rough draft designs are available, the use of kinematic analysis is very useful. Using the Simcenter motion application, principle sketches or simple curves can be moved and their dimensions can be optimized. Thus, the sketches of the early design phase become movement-based control sketches. In the further design process, the kinematic models can be used repeatedly to check the latest state of the mechanism.

As soon as mass properties are assigned to the CAD geometry, motion analysis can be extended to dynamic analysis. In this case bearing and contact forces, velocities and accelerations can be determined. Therefore, motion analyses are often the basis for subsequent FEM analyses because FEM uses bearing forces as boundary conditions. Based on

Mass properties of the components expand the area into the dynamics.

the results (forces and displacements) it is possible to choose springs, dampers, additional masses, vibration absorbers, bearings (load capacity) etc. from supplier catalogs.

Users of Simcenter motion should have experience in modeling components and assemblies with NX. This is necessary because some examples in this chapter are not only based on finished assemblies, but also partially intervene in the design methodology. However, no further previous experience is required.

Simcenter motion covers the part of the mechanics that deals only with rigid bodies. Usually there is a plurality of rigid bodies that are connected to each other by joints. Such problems appear, for example, in chassis of motor vehicles. The software for the analysis of such tasks is described by the term *MBD* program (*Multibody Dynamics*) (Figure 2.1).

Subdivision of technical simulation into four fields

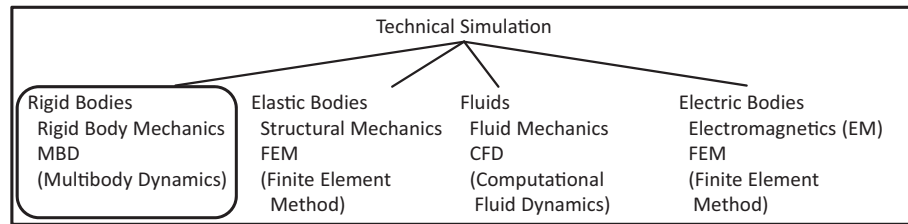


Figure 2.1 The four common fields of technical simulation

Within the CAD model, the user defines moving rigid bodies (links), joints, drivers, and possibly external forces or constraints. Even springs and dampers may be involved (Figure 2.2).

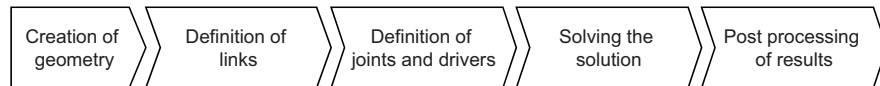


Figure 2.2 Steps to process a MBD analysis

Links are usually defined using CAD geometry (components and assemblies). In addition, the CAD system, with its powerful capabilities, can also be used to define, for example, cams or other control elements.

2.1.1 Simulation Methods

Additional literature

It is difficult to generalize how MBD methods work because the different solvers, including RecurDyn and ADAMS, work quite differently. For a detailed description on ADAMS, see [adams1]. For a detailed description on RecurDyn, see [RecurDyn1]. The current solver which is used by default and supports the new functionality is NX Motion/Simcenter Motion. This solver is documented in the NX Online Help. For the examples in this chapter, we use the NX Motion Solver and for the Co-Simulation example RecurDyn is used.

Internally, the moving bodies, joints and drivers are converted into a mathematical system of differential equations, which is solved to determine the desired quantities (Figure 2.3). This includes the displacements, velocities, and accelerations of the moving bodies and joints, as well as the reaction forces at the joints.

Each component or geometry that should be used as moving body has to be cut free. Six dynamic equations (describing the relation between forces and accelerations) and six kinematic equations (describing the relation between positions and velocities) in the translational and rotational directions are set up. All of these equations together form a system of equations describing the mechanism and its motion.

The number of unknowns in the system of equations can be reduced by adding constraints. Each joint that restricts the possibility of movement of one or between two bodies could be expressed in the form of additional equations in the system of equations. For example, a revolute joint between two moving bodies leads to a reduction of five unknowns in the system of equations because only one rotational degree of freedom remains where once there were six.

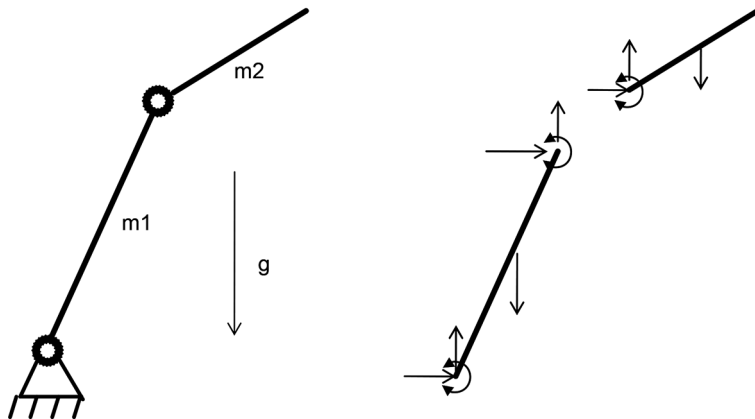


Figure 2.3 A differential system of equations is set up.

Motion drivers, which define displacement, velocity or acceleration, also reduce the degrees of freedom (DOF). A rotational driver, for example, with an enforced speed of 360 deg/sec, reduces the number of DOF by one. On the other hand, forces and torques, appearing on the motion model, neither bring additional unknowns into the system nor reduce the count of DOF.

That way, the count of DOF is reduced either to zero (in which case the system of equations can be solved directly) or to a number greater than zero. In the second case, the system can be solved by adding initial conditions and integrating the equations over the time. In the case of zero degrees of freedom, we have a kinematic system; otherwise, we have a dynamic system to be solved.

It also should be noted that the resulting system of equations is either linear or non-linear, depending on the correlations that the various types of joints introduce in the system. While simple types of joints such as revolute, slider or spherical joints behave linearly,

Drivers and constraints reduce the number of unknowns.

Some kinds of joints cause non-linearity in the system of equations.

complex joints such as the *point on curve* connections require non-linear equations. Linear equation solvers – as they are usually used for FEM – are therefore not used for solving MBD systems. For MBD systems, solvers are used with the ability to reduce the order.

After solving the system of equations, the following variables are available for post-processing:

- translational velocity
- rotational velocity
- coordinates of center of gravity
- orientation angles and coordinates of position
- applied, external forces
- forces in joints and constraints

2.1.2 Restrictions

Restrictions of MBD systems and demarcation to FEM

A very basic property and restriction of MBD is given by the rigidity of the considered bodies. A moving body (rigid link) can be moved in space, but cannot be deformed. For MBD, real bodies are reduced to their mass and inertia properties and their geometrical dimensions, while their deformation properties are neglected. This is the basic difference to structural mechanics, which uses FEM to consider flexible bodies, including their stiffness and so their deformations and stresses. The disadvantage of linear FEM compared to MBD is that no movements and only small deformations can be simulated. Therefore, the assumption of rigidity in the motion links in MBD has the advantage of simplifying the analysis and reducing the computational effort, thus enabling even complex motions of large assemblies to be analyzed.

Clearances, tolerances and flexible bodies can be modeled in MBD only with greater effort.

In reality, however, there are some effects that are difficult to model using MBD. These include clearance, tolerance, and flexibility. Because such effects are often not taken into account in the MBD model, in some cases it may appear, for instance, that a clamping situation has occurred, when in reality there is a slight clearance in the joints or there is some flexibility in the body to ensure motion without any problems.

Clearances can be considered in MBD as well, but then the corresponding parts must be considered dynamically and the contacts with restoring forces must be modeled. If so, the system will have unconstrained degrees of freedom, which will make the problem significantly more difficult to define and solve.

2.1.3 Classifications of MBD

Classification of dynamics

For a classification of motion simulation we refer to the classification of mechanics as, for example, it is described in [HaugerSchnellGross]. Accordingly, the mechanics may be divided into *kinematics* and *dynamics*.

Kinematics is the science of the temporal and spatial movement, without regarding forces as a cause or effect of the movement. The *dynamics*, however, deals with the interaction of forces and movements. It is divided into the *statics* and *kinetics*. The statics deals with the forces at stationary bodies e.g., a truss in equilibrium), while the kinetics examines actual movements under the effect of forces (Figure 2.4).

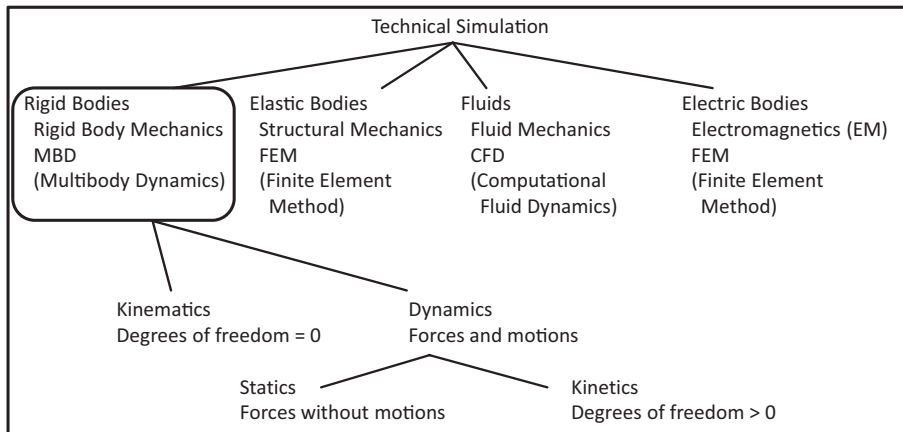



Figure 2.4 Classification of MBD Simulations

All these phenomena can be analyzed with NX/Simcenter Motion, with the general restriction of rigid bodies in MBD systems. However, starting with version NX 7.5, it also is possible to take into account single flexible bodies in the MBD system. These bodies must be prepared in advance using FEM, which means that the stiffness matrix must be determined (in reduced form) and included as *Flexible Body*  in the MBD system.

Flexible bodies are a special case.

Kinematic systems are characterized by the fact that all degrees of freedom (DOF) of a moving body are determined. This determination may be made either by joints or by driver rules. Such systems are predictable to a certain extent, and can also be referred to as motion-driven systems (tied movement).

Determined and undetermined degrees of freedom

Kinetic (Dynamic) systems are available if one or more degrees of freedom are undetermined. In this case, the motion results from external forces (untied movement). For example, the force of gravity can lead to a swinging movement of a lever with rotational DOF. Kinetic systems are therefore also known as power-driven systems.

■ 2.2 Learning Tasks for Kinematics

2.2.1 Steering Gear

This basic example will explain the most important technical matters that are necessary for a simple motion analysis using Simcenter Motion (Figure 2.5). The example will take the user through the process of generating links (the motion bodies in Simcenter 3D) and basic joints and uses the *Articulation* function as the driver since it is well suited for purely kinematic motion simulations. In addition, the function for dynamic analysis will be used as a method for detecting unconstrained degrees of freedom.

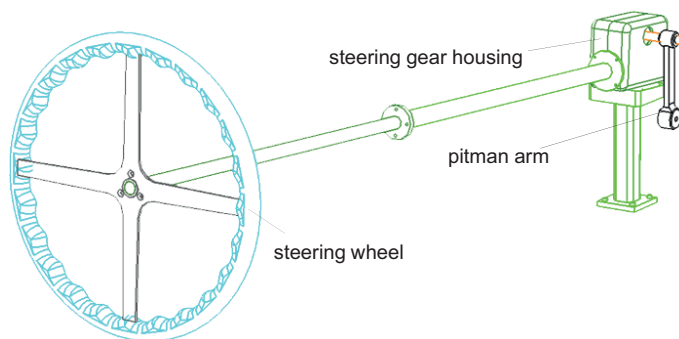


Figure 2.5 CAD model of the first example of motion simulation

This kinematic model based on a subassembly first is defined as a stand-alone mechanism. In a later example, this mechanism will be modularly assembled with other mechanisms in a higher level assembly to form a larger motion model.



This basic example should be performed by everyone who wants to work with Simcenter Motion.

2.2.1.1 Task

The goal is to control the design.

A designer has redesigned the levers for the steering gear. Now the designer has to check if collisions occur in the mechanism. Therefore, a kinematic model must be created that allows the rotational movement of the steering wheel, and (associated with it) of the stalk on the steering column.

In this task, the steering gear of the RAK2 and its steering wheel and the control stalk are used. The steering gear is accommodated in a housing and connects the steering wheel with the control stalk.

For this task, the simulation should only be used for visual control. However, the examination of minimum distances to other components, the study of the resulting reaction forces in the joints and collision checks would be possible in further analyses.

In the following section, at first some principles are explained. Thereafter, the necessary solution steps for this example are presented. Very hurried readers can skip the next section and move straight to the creation of the model (see Section 2.2.1.4).

2.2.1.2 Overview of the Functions

In the *NX/Simcenter Motion* application the kinematic or kinetic model is set up and the simulation is performed and evaluated. Figure 2.6 shows the *Motion* toolbar that appears after changing to the *Motion* application. The *Home* toolbar and the *Analysis* toolbar contains all the main features of the *Motion* module that are needed to set up, solve and post process motion analyses.

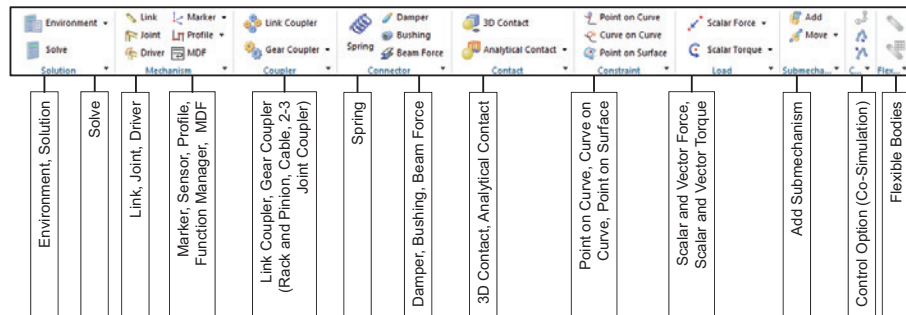




Figure 2.6 Overview and brief explanation of the *Home* tab functions of *Motion*

In the subsequent section we give an overview of the main functions in the *Motion* module with a reference to later use. Very hurried readers can skip this section and immediately move on to the creation of the model (Section 2.2.1.4).












-  The function *Environment* allows to define the basic setting of the system for kinematic or kinetic properties (herein called dynamics). For our task, we will use the “dynamics”, although it is actually a kinematic model. The reason for this approach is, that the user has more possibilities, which contribute to a better understanding and error identification. In addition, in the *Environment* dialog advanced solution options can be selected for the RecurDyn solver. The option *Component-based Simulation* is suited for assemblies, because it activates the selection filter for assembly components for the generation of *Links*.
-  The *Solution* function must be activated by the user to specify the type of solution that is desired. The options include the *Normal Run*, the *Articulation*, and others. In addition, the gravity and the solver parameters are defined here.

Defining and solving a motion solution

For running the analysis, first a solution has to be created. An existing solution could be calculated with the following function:





-  *Solve*

Links and joints to set up the motion model

-  The main elements for the definition of the motion model are the rigid moving bodies called *Links*. With this function the user defines which geometry or component should be a part of the mechanism.
-  In addition to the links, the user defines joints, which specify how the connected links can be moved by constraining DOF between two links or between one link and the environment. In this context the function *Driver*  is also used, which is necessary to drive the joints. If the *Joint* function is opened, you can find a lot of different joint types as a selection. These are the most important joints:
 -  The *revolute* joint only allows a rotation about a defined axis.
 -  The *slider* allows a translational displacement between two parts or one part and the environment.
 -  The *cylindrical* joint allows the rotational and translational displacement along one axis.
 -  The *screw* forces a rotation if a part is displaced in translational direction along the axis of rotation.
 -  The *universal* joint allows tilting movements between two parts. However, a rotation around the main axis is transmitted to the other part. Depending on the angular position of the axes it can cause uneven rotational speeds as in real universal joints. This non-uniformity can be avoided by use of the *constant velocity* joint described below.
 -  The *spherical* joint allows all rotational movements and constraints all translations.
 -  The *planar* joint allows the frictionless sliding of two parts in a plane. The rotation about the normal direction is also possible.
 -  The *fixed* joint eliminates all degrees of freedom so that there is no displacement between a part and the environment, or between two parts. In addition, the fixed joint could be modified in a way that only the desired DOF are fixed. So it could also be used as some user-defined joint.






The joint primitives

In addition to the conventional joints, which are based on realistic joints, there are some *joint primitives* that offer more precise control of the DOF of the connected links. So it is possible to fix every single degree of freedom with the help of joint primitives. Here you can find some of the useful joint primitives:

-  *Constant Velocity*: This joint works very similar to the universal joint described earlier. But unlike the universal joint, the rotational velocity is constant on both sides and even angles with over 90° are possible too.
-  *Inline* fixes two translational DOF, so that the both links could be moved on one axis to each other (similar to *Point on Curve*). All rotations are free.
-  *Parallel* is a joint that keeps two faces, lines or axes in parallel. Two rotational DOF are fixed. All translations are free.
-  The *Orientation* primitive fixes all rotational DOF between two links but allows the translation in all three directions.






Further functions of the main *Motion* toolbar are:

Sensors etc.

-  *Smart Point*: A general CAD point that is associative to the geometry it was assigned to.
-  *Marker*: A marker that is used to request results such as the velocities and accelerations on certain positions of the link.
-  *Sensor* enables the user to record motion results such as displacements, velocities, and accelerations relative to other results or markers.
-  *Profile*: A Profile could be used to define a driver based on a sketch curve.
-  *f(*)* The *Function Manager* is used to define and manage functions, for example to control a driver time- or motion-dependent. Some simple functions, however, are available directly in the appropriate motion features. Therefore, the function manager is not needed in such a case.






Another group of special joints are couplers and gears. The user can choose between the following options:












Couplers

-  *Link Coupler*: With the *Link Coupler*, several Links can be coupled directly according to their Degrees of Freedom.
-  *Gear Coupler* defines the relative motion of two revolute joints or a revolute and a cylindrical joint with a defined ratio.
-  *Rack and Pinion* defines the relative motion of a revolute and a slider joint with a defined ratio.
-  *Cable* defines the ratio of the relative translational motion of two slider joints.
-  *2-3 Joint Coupler* defines the relative motion between two or three revolute, slider and cylindrical joints.

The next group of motion objects summarizes the connections:

The connector elements are collected in a *Home* tab group.

-  *Spring*: Flexible element that is defined between two joints, or a joint and the environment or on an existing joint with stiffness value, preload and damping coefficient.
-  *Damper*: Damper element defined like a spring, but with a damping coefficient. This results in a velocity-dependent force between the respective links.
-  *Bushing*: A cylindrical combination of spring and damper (stiffness and damping coefficient in all directions).
-  *Beam Force* defines a connection with section area, inertial properties and material information to calculate the stiffness matrix and the resulting forces based on simple linear beam theory.
-  The *3D Contact* is a function to calculate the impact and take off of the defined solid contact partners and the resulting forces. Actually these contact definition is no joint but a force based object that responds by restoring forces in the event of contact. In this case, friction and contact damping could be taken into account too and could be calculated using assigned parameters. The contact should be used with caution due to the numerical complexity (long solving times in certain circumstances). If possible, one of the following alternatives should be used (*Analytical Contact* or constraints).

- Various types of constraints
-  **Analytical Contact:** Analytical calculation of contact behavior with simple substitutional geometries like *Sphere to Sphere* or *Extrusion to Revolution* or some other combinations.
- Now a group follows, in which the constraints are summarized. These include:
-  **Point on Curve** forces a point on a link to move along a desired curve.
 -  **Curve on Curve:** Two curves are forced to slide tangentially on each other. Both curves have to be coplanar. With this function most of the cam disc tasks are solved.
 -  **Point on Surface:** A point on a link is forced to slide on a selected surface.
- Forces and Torques
- Additional motion features in the toolbar belong to the group of loads. These include:
-  **Scalar and Vector Force**
 -  **Scalar and Vector Torque**
- Submechanism
- In NX 12 there is a new option to use existing mechanisms defined on subassemblies in a higher level assembly called submechanism. The included submechanisms are associative to their source mechanism.
-  **Add**
 -  **Move**
- Some advanced features are only available after appropriate adjustment of the settings in the environment dialog. These include:
-  **Control Input** defines the control variables that are read from the optional Matlab Simulink control and which are provided to the MBD model, for example as a driver.
 -  **Control Output:** Measured value, which is fed to the Amesim or Matlab Simulink loop.
 -  The **Flexible Link** function allows calculated flexible links, which previously had been calculated with FEM, instead of solely using rigid bodies.
- Now the solution can be solved, and the results are available after the solving process has finished.

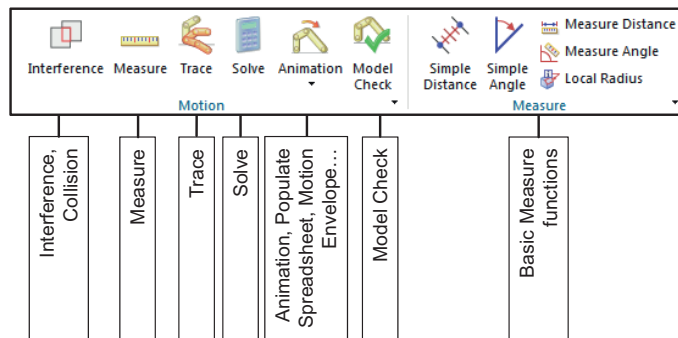











Figure 2.7 Overview and brief explanation of the *Analysis* tab functions of *Motion*

Among others, the Analysis tab provides functions for geometric analysis. This includes the following functions:




-  *Interference* checks the model for collision and could create intersection solids.
-  *Measure* for measuring distances and angles
-  *Trace* for recording the geometry during the movement

The drop down list under the animation button provides additional function for post processing:

Assistance for post-processing

-  *Animation* displays the calculated movements of the model.
-  *XY Results* for the graphical evaluation of motion results
-  *Populate Spreadsheet* for editing, re-using, and saving of motions using a spreadsheet as input
-  *Create Sequence* saves a motion animation in an assembly sequence so that the motion sequence is also available in the master assembly.
-  *Load Transfer* for transferring reaction forces from the kinematics analysis in the FEM application
-  *Motion Envelope*: Creates a lightweight body of the volume that is coated by the motion of the selected solid bodies. Could be recorded with an absolute reference or relative to another link.

2.2.1.3 Overview of the Solution Steps

To solve this exercise, a Simcenter motion file must first be created in the NX system. Then the geometry that should be part of the motion mechanism has to be defined with the *Link*  function. The creation of two revolute joints, a gear coupler and a driver on the steering wheel, follows. The time-dependent *Normal Run*  solution is used to find accidentally unconstrained degrees of freedom, and the *Articulation*  solution is used to manually move the driver on the steering wheel.

The steps of the exercise

2.2.1.4 Creating the Simcenter Motion File

According to the master model concept, all elements used for motion analysis (links, joints, drivers...) are stored in a separate file (i.e. the kinematic sim-file; see Figure 2.8). This sim file is connected to an assembly file (or a prt-file) via an assembly reference. This means that the kinematic file is a quasi-assembly with the assembly which should be analyzed as a single component. In addition to these assembly references, other associative connections are added to reflect the associative relationships between the joint and link objects defined in the kinematics file with the geometry objects describing these components (curved arrows in Figure 2.8). So the *NX/Simcenter Motion* application is fully integrated into the master model concept, similar to the *NX Drafting* application.

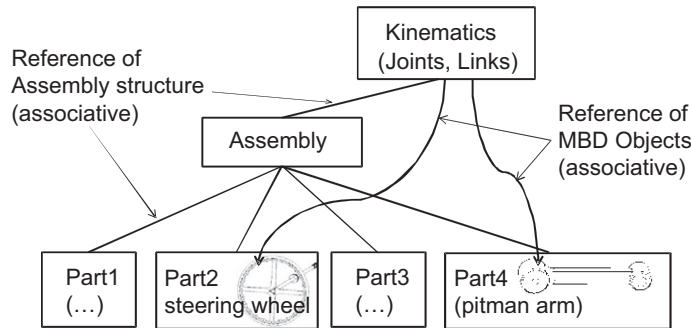


Figure 2.8 With the help of the master model concept, the entire product is digitally mapped. Internal references between geometry and MBD joints are created.

The exercise starts here.

This motion structure is automatically created when the user creates a simulation as shown here:

- 1 Load the assembly from which you want to create a simulation in the Simcenter 3D system. For our exercise, the assembly file *ls_lenkgetriebe.prt* should be opened. The files are located in the *RAK2* directory of the downloaded data.
- 2 Start the *Motion* application.

The *Motion Navigator* appears as the first tab in the resource bar. This navigator supports working with the *Motion* application by representing all defined features and providing options to edit them.

The navigator shows the structure of the model and allows the manipulation of the features.

The Motion Navigator only shows the master node, i. e., the model that you have opened (see Figure 2.9).

- 3 Create a first simulation by right-clicking on this master node and select the function *New Simulation* in its context menu.

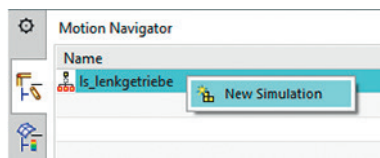


Figure 2.9
Motion Navigator with master part

In the occurring *New Simulation* dialog the name of the sim-file and the folder to store it can be checked and modified.

- 4 Keep the suggested name and check if the folder is in your work directory. Accept with OK.
- 5 Confirm the following menu *Environment* by selecting OK. We will come back to it later on.

After activating this function, the system creates a simulation file that is associated with the master model via the assembly structure. In addition, the function *Motion Joint Wizard* is activated automatically, which tries to create *links* and *joints* according to the existing assemblies *Mating Conditions/Constraints* (Figure 2.10).

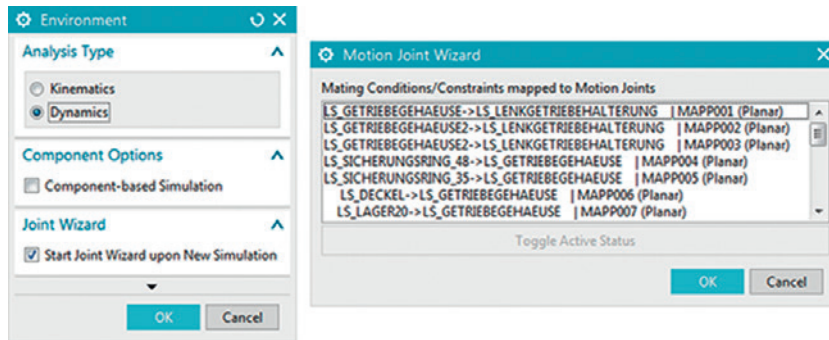


Figure 2.10 The Motion Joint Wizard implements the Mating Conditions/Constraints into motion joints.

The *Motion Joint Wizard* function analyzes each set of constraints with respect to the degrees of freedom that exist between the affected assembly components. If there is only one undetermined rotational degree of freedom, then a *Revolute* joint is generated. If there is an undetermined translational degree of freedom, then a *Slider* joint is generated. An assembly constraint that positioned, for example, one point to another, is translated by the *Motion Joint Wizard* into a *Spherical* joint. An assembly constraint that defines all degrees of freedom between two parts, is translated into a fixed joint. In a similar manner, a few more joints can be generated automatically.

The *Motion Joint Wizard* can therefore automatically create the motion model, or parts of it, when the assembly on which the motion should be based on has been constructed in a way that the mating already describes the potential movements of the components. This approach can be quite useful, but the following disadvantages must be considered:

- The *joints* automatically generated by the *Motion Joint Wizard* are not associative to the geometry. That means that in case of changes on the master model the joints must be adjusted manually. A manual creation of the associativity of the joints is subsequently possible.
- Only the assembly constraints of the top-level assembly are analyzed and converted. Constraints from the subassemblies are not considered.
- Assembly constraints are often used for parts that are irrelevant in terms of the motion model, such as small bolts, nuts and washers. In the case of automatic translation by the *Motion Joint Wizard* all these parts are created as motion links. In this case the motion model is considerably more complex than it needs to be. One remedy for this problem is to disable single conditions in the *Motion Joint Wizard*.

For these reasons, the *Motion Joint Wizard* should not be used for the solution of our problem:

- ✚ Cancel the *Motion Joint Wizard* with the CANCEL function.

The Motion Navigator should now have a structure as shown in Figure 2.11.

Advantages and disadvantages of Motion Joint Wizard

We are not using the Motion Joint Wizard.

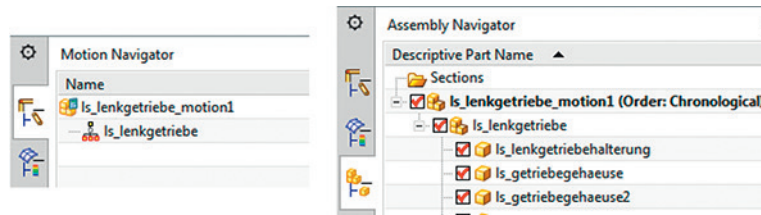


Figure 2.11 Motion and Assembly Navigator after creating the motion sim-file

The motion model is a quasi-assembly of the master model.

So the NX system has automatically created a new file that is associated with the master model according to the master model concept. The Assembly Navigator can now be used to represent or to work with the new structure. Figure 2.11 shows the Assembly Navigator to the right of the Motion Navigator, which now displays the motion model as the top-level assembly.

Furthermore, you should check in which operating system directory the new file was stored. To do this, display the folder in which the master file can be found in Windows Explorer. Figure 2.12 shows on the left side the master file *ls_lenkgetriebe.prt*, which can be located in any folder of the operating system. Once a motion model is created, the NX system creates a subfolder with the name of the master model. All the data that is needed for Simcenter motion is stored in this new subfolder. In our case, we see that the folder now includes *ls_lenkgetriebe_motion_1.sim*, which is the file for the motion model.



Figure 2.12 The resulting files of the simulation are stored in a folder.

During the following simulation several additional files are created, which are all stored in this folder as well.

2.2.1.5 Selection of the Environment

As a next step, the environment for the motion model should be adjusted. This is done by using the *Environment* function. There are two alternatives, *kinematics* and *dynamics*, which correspond to the classes of mechanics that were described at the beginning of this chapter.

The following should be observed for the use of these two classes in the NX Motion application.

Kinematics

The decisive factor is whether unconstrained degrees of freedom have to be processed.

The key characteristic of kinematic analysis is that all the degrees of freedom for the entire system are constrained. In this context, each *link* can have three translational and three rotational degrees of freedom. Hence, if a kinematic simulation is to be performed, the user must ensure that none of the *links* can freely be moved. Instead, all movements must be determined by joints, constraints or drivers.

Of course, no conflicts may arise from joints or drivers in the mechanism. Overdetermined DOFs cannot be processed by the solver, so all redundant degrees of freedom are fixed automatically. This can lead to unexpected results and errors though. Experience has shown that in the most cases large kinematic models will only work correctly if they are defined without redundancy. Smaller models, however, will usually run with limited redundancy without any problem but should also be avoided.

Overdetermined and redundant degrees of freedom

The advantage of the kinematic environment is that no mass properties are required for the *links*. The disadvantage is that the user is forced to create a motion system with exactly zero degrees of freedom. Until this is created, it is not possible to perform a test run.

Dynamics

Dynamic analysis is characterized by the possibility of undetermined degrees of freedom and free movement opportunities. Such movements are obtained by including the mass and inertia properties of the *links* as well as the external forces such as the gravitational acceleration in the analysis.

Dynamics also allows the simulation of undetermined degrees of freedom.

A dynamic analysis will calculate results even if undetermined degrees of freedom are available, while a kinematic analysis will stop in such a case. This is an advantage for dynamic analysis during the model-building phase when the joints have not yet all been defined. To do this, the mass properties for each *link* must be assigned and verified.

For these reasons, the dynamic environment should be chosen for the solution of our problem, even though no undetermined degrees of freedom are desired. We use this method only to simplify the development of the motion model, so that we can temporarily test the model without having fully determined degrees of freedom. Once the model is complete, we can then easily switch back to the kinematic environment.

In addition, *RecurDyn Solution Options* can be selected in the environment settings. These include the *Motor Driver*, which defines an electric motor based on its electrical parameters and for which a signal diagram can be submitted; the *Co-Simulation*, which allows to couple controls that have been defined to the NX-motion model using Matlab Simulink; and *Flexible Body Dynamics*, with which it is possible to work not solely with rigid moving bodies, but to also make them partially flexible. For this, however, a prior FEM analysis of the corresponding parts is required. By working with Simcenter Motion, it is not necessary to define such settings in the environment dialog.

Description of the advanced solution options

Furthermore, you can choose whether a *Component-based simulation* shall be used or not. This is useful if assemblies shall be simulated. With this option, the link selection filter is preset to components. However, this can always be changed manually. A component based simulation model is necessary if assembly functionality should be used out of the simulation, e. g. if arrangements should be created from specific mechanism positions.

2.2.1.6 Definition of the Links

Now the links will be created.

- ✎ Select the *Link*  function.

First, a link should be defined to describe the steering wheel, and then a second one to describe the pitman arm (Figure 2.13).

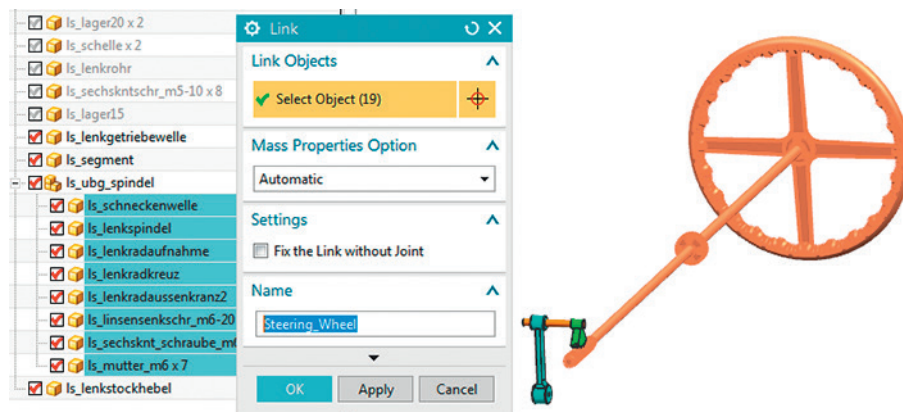


Figure 2.13 The menu for the definition of a link

Assembly components as well as solids, simple curves, points and datums can be moved.

The first selection step concerns the selection of the geometry that should be part of the link. If you have not already selected the setting to filter for components in the environment, it should be set now. This will make it easier to change the geometry of the assembly components later without the danger of the links in the motion model losing their references.

- ✎ Now select the assembly components that belong to the steering wheel in the graphics window; that is, all the parts that move together with the steering wheel.

19 components belong to the sub-assembly *ls_ubg_spindel*. You can use the Assembly Navigator to select these components.

Mass properties are determined automatically for solid bodies.

Once you have selected the components, the mass properties of the link can be defined as shown in the following selection steps. However, this is not necessary in this case because the system can automatically calculate the mass properties based on the geometry and the assigned material respectively the density. Because these properties are not of interest in this exercise, we will use the automatic mass analysis instead, which is set as default. Therefore, keep the *Automatic* setting under the *Mass Properties Option*.

- ✎ In the field “Name” enter an appropriate name, such as “steering wheel”.

Do not use any spaces or special characters in the name of motion objects.

- ✎ With a click on OK or APPLY, the link will be created and displayed in the Motion Navigator in the *Links* group.

- ✎ Now, in the same way, create the next link. Add the three components *ls_lenkstockhebel*, *ls_segment* and *ls_lenkgetriebewelle* and then name the link *Pitman_arm* (Figure 2.14).

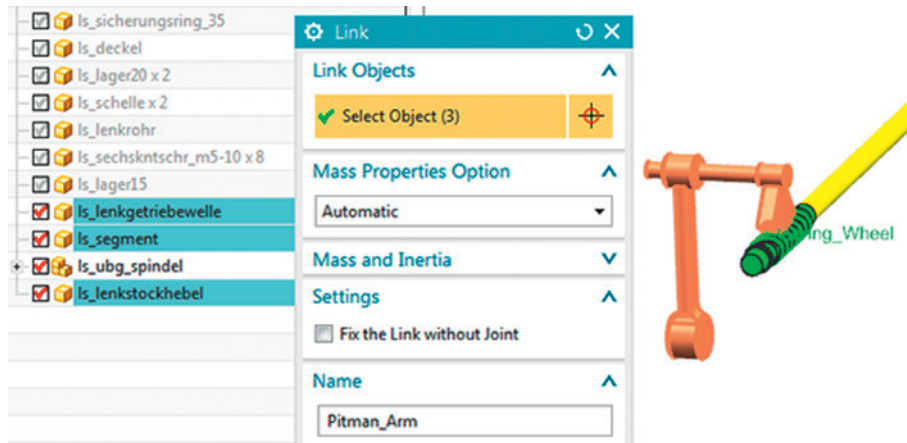



Figure 2.14 The definition of the Link for the transmission joint

At this point, all the necessary links have been defined. In the next step we will define the joints.

2.2.1.7 Definition of Revolute Joints

Now we will define a revolute joint between the link steering wheel and the fixed environment. Other joints can be defined similarly. Proceed as follows:

✎ Select the *Joint*  function. You will see the menu shown in Figure 2.15.

At the top of the menu the type of the desired joint can be selected. The default is the *Revolute*  joint, which is a joint that has only one rotational degree of freedom. Since this is the desired joint, we will proceed with the selection steps.

With the first selection step the first *link* which shall be connected (in this case the steering wheel) is specified. In principle, the steering wheel can now be selected in any manner in the graphics window, but it is advisable to take into account the following aspects for the selection:

- It is recommended to select a geometry from which the system can derive the desired joint center and the axis of rotation. This is, for example, possible with a circle: In this case, the circle center becomes the center of rotation and the circle normal becomes the rotational axis. Also a straight edge or curve is possible: In this case the next control point becomes the center of rotation and the direction of the edge or corner becomes the axis of rotation.
- It is also advisable to select a geometry that is subjected to changes as few as possible in the further design history because if the selected geometry is subjected to changes, it is not sure if the joint remains associative to the geometry and is updated automatically. For example, if an edge is selected, which is rounded later, the joint loses its associativity to the geometry.

For the definition of revolute joints the selection should be done on arcs. Then the point and axis of rotation can be determined automatically.

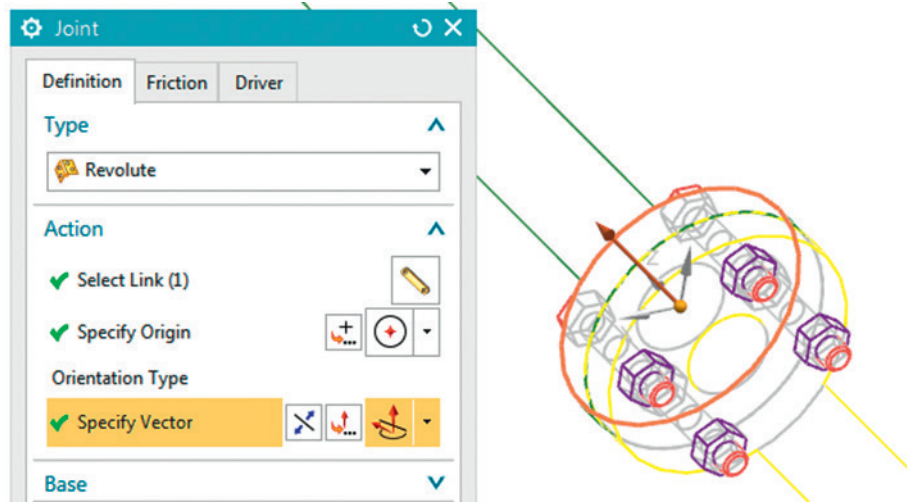


Figure 2.15 The best way for Revolute joint definition is to select circular edges. Then, the origin and vector can be determined automatically.

- Therefore, select a circular edge on the steering wheel which is not subjected to significant changes.

A joint can often be generated with just two mouse clicks.

In the second step, the origin and orientation of the joint should be selected. In the case of the revolute joint this is the center and the axis of rotation. Because these two pieces of information have been given in the first selection step, this question does not need to be answered.

In the base section of the dialog the second link which should be connected can be selected. If nothing is selected the system assumes that the joint connects the first link to the fixed environment. Because this is desired here, no selection is made in this step.

- Accept with OK to create the revolute joint.

In the graphics window and in the Motion Navigator the joint is now displayed (Figure 2.16).

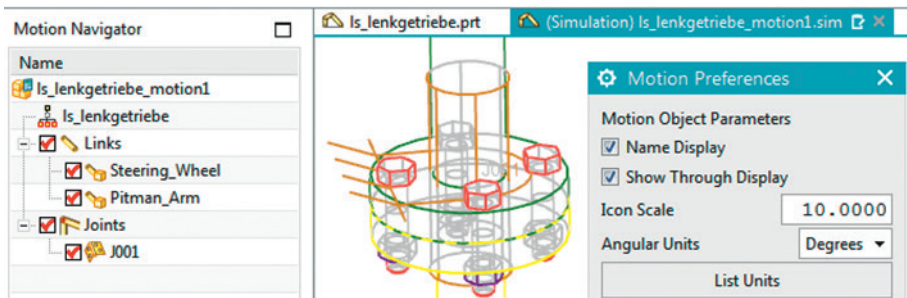


Figure 2.16 A joint which is connected to the fixed environment can be identified by its symbol.