1. ADVANCED TOOLS OF NX SYSTEM IN PRODUCTION ENGINEERING

1.1. Modeling using the synchronous tools

In the course of this project the students will get familiar with:

1. Identify the difference between working in History Mode and History-Free Mode.
2. Modify faces using specific distances and angles.
3. Edit blended and chamfered corners.
4. Copy and paste faces.
5. Control the location of faces using dimensions.
6. Specify geometric relationships between faces.

Initial Requirements - The only requirement for this course is the basic knowledge of NX Modeling.

Modeling modes - While working in the modeling application it is possible to be in one of two modes:

- History mode.
- History-free mode.

History mode

 In history mode the user may create and edit a model using an ordered sequence of features which are displayed in the Part Navigator panel. It is the main mode adopted in NX designing based on a traditional approach to modeling in terms of the historicity of modeling. This mode is useful for highly-engineered parts subjected to modifications. It is also useful for parts assigned to modifications at a further stage of designing, with the use of predefined parameters that are built into the sketches, features, and feature order used to model the part.

History-free mode

 History-free mode is useful when the user has to explore previously established design concept and there is no need for planning next stages of modeling in advance. It may also be useful for further modifications. In this mode it is possible to create and edit a model on the basis of its current state without an ordered sequence of features. It is possible to use the same commands that are available in the History mode. In this mode only the local features that are not reliant on the sequential structure are created.

1.2. Definitions of programme groups, tool groups, geometry and NX methods

During this project students will familiarize themselves with:

1. Defining new tools and the way in which it is possible to retrieve tools already existing in the NX's library.
2. Creating and using Machining Method objects group to organize operations according to cut method.
3. Creating groups of different types of geometry.
4. Creation of program objects in order to group them within logic programs via clean-up actions.

Tool objects

In this section, the students become familiar with the rules of creating NX tool objects. They learn the principles of defining new tools and methods of downloading new tools that already exist in built-in internal system library.

Objects from Machine Methods group

Machine Methods aim to enable proper organization of procedures in accordance with the philosophy of technology, grouping them into appropriate compartments.

This allows for the grouping of roughing treatments, finishing and semi-finishing treatments in one place by giving them the same technological parameters in terms of industrial treatment surplus and tolerance of the generated machining tool paths.

Objects from Geometry group

Geometry group is used to define the areas of a part to be processed and to allocate an appropriate orientation of the part on the lathe to the technology being designed. There are also defined parameters such as: the workpiece geometry, the geometry of the finished part, omitted geometries, the orientation of the machinery in respect to the part's geometry, and surfaces safe for the courses of the tool.

Depending on the type of operation which defines the given procedure, it is required that another solution in the selection of the type of geometry is adopted. For planar milling it is necessary to define the geometry of the processing boundaries, while for contour operations of groups of processed surfaces or the whole geometry parts.

Objects from the Programme group

Programme objects allow for gathering and organizing operations in the programming in a logical way. For example, one program may consist of the operations defined on the top side of the part, whereas another program may consist of the operations located on one of the side walls of the part.

1.3. Elaborating the documentation concerning the milling treatment technology with the usage of technological templates.

Technological templates are designed to guide the user in a complex way through a typical course of a technological process, which is required to create different editing programmes. Templates are particularly useful for the beginners, because they have an easy to use interface which allows on a step by step basis for creating objects and determining parameters of each operation.

Through the use of the Shop Documentation command  it is possible to generate customized information in the form of reports that can help in the process of manufacturing parts. Such information may include: tools and the material of the workpiece, control geometries, machining parameters, and post for operators, information concerning paths of tools. For this purpose it is possible to obtain information both in the form of ASCII code, as well as in the form of HTML code.

In the course of this project the students will familiarize themselves with:

1. Creating a basic rest milling program using the Manufacturing wizard (template).
2. Creating technological documentation which may be used by machine operators.

1.4. Capabilities of high speed machining (HSM) in NX and automation of machining based on features

In NX Manufacturing, any shape that is recognised as suitable by the software may be a feature of machining. In the course of this project the students familiarize themselves with:

- The principles of creating machining software based on machining features.
- Improving the machining efficiency by optimizing programs based on machining features.

Machining features automate the creation of technological operations. In NX Manufacturing any shape being recognized as suitable by the software may be a feature of machining. NX combines a selected feature with a set of instructions necessary to manufacture this feature. Machining features may be selected for geometries such as:
geometries of holes, grooves, sockets, pockets, and features defined by the user for irregular shapes.

In order to perform the automation of creating machining process, NX uses the standard knowledge of machining to:

- group similar features,
- create appropriate operations,
- select an appropriate tool,
- determine the method of machining and its parameters,
- optimise the tool's path.

1.5. Implementation of lathe treatments in NC CAM

In the course of this project students familiarize themselves with:

- Defining parameters that are commonly used during turning operations.
- Initialize a turning setup.
- Defining the geometry of the workpiece and of the semi-finished part.
- Specify cut region containment.
- Creating an operation that uses various parameters of machining.
- Generating the tool paths.
- Creating and retriev tools from NX library.

1.6. Assistance of design and manufacturing of injection mold type of components using Mold Wizard

NX Mold Wizard is a set of tools to assist in the process of designing injection molds, in particular:

- Counteracting the defects of moldings (i.e. warping) at the stage of mold conception.
- Analysing the technological aspect of the molding and carrying out a simulation of injection.
- Constructing the parametric 3D model of the mold.
- Verifying the construction of the mold.
- Carrying out the kinematic simulation of the working mold.
- Automating of the process of generating the 2D documentation (assembly and implementation)
- Exploiting the existing models of molds in new projects.

Mold Wizardapplication was developed and tested with the continuous participation of mold designers and manufacturers. This ensures that the application meets the needs and priorities of mold constructors in response to design challenges in the real world.

Mold Wizard has the tools and procedures used in order to automate the processing tasks of difficult and complex mold constructions. The application allows for saving time during the entire process of designing and ensures receipt of full 3D models ready for production. If any element of the designed construction changes, it does not imply a lot of work and time because changes in the model are combined with forming elements.

During this project students will learn how to create a tooling mold assembly for a simple model of parts, and will perform some necessary steps in order to complete the installation of the mold assembly.

The students will, among other things:

- Validate the model in terms of its moldability.
- Initiate the project.
- Define the coordinate system for the mold.
- Define the injection mold.
- Perform the steps towards the parting process.
- Add the mold base.
- Add sprue, bushings, ejector pins and other standard elements.
- Add pockets.

1.7. Designing of welding processes with the use of Weld Assistant tools

This project will familiarize the students with issues related to designing of welding processes, in particular arc welding of welds, welding spots, adding annotations to the drawings, and handling tools used to identify groups, and assigning tolerance to welding processes.

In the course of this project the students will learn how to:

- Change the default setting system for welds.
- Change the preferences of the system for welds definitions.
- Create several types of welds.
- Automatically add to the drawing endorsement about the weld.

Functions of Weld Assistant are used to automate creating and managing functions of welding on 3D CAD models. By using the Weld Assistant it is possible to:

- Create, edit and delete weld elements from the model.
- Import welds from other projects saved as a neutral format.
- Welding spots can be exported to the neutral format.
- Review the deployment of welds' elements and to identify any compound that violates the standards set by the Advisor type of the weld.
- Adjust function of weld control based on custom practices and standards, using the default custom settings of the client.
- Interpret functions of the weld and create a suitable weld or weld symbol on the drawing, using the function of automatic creation of descriptions.
- Generate requests for additional information such as ID unit, ID of group, class and tolerance which are saved with the feature.

Publish welding characteristics to Teamcenter as single, individual objects, allowing them to be managed and processed by the environment used to production managing in the form of Teamcenter and Teamcenter Manufacturing software.

1.8. Programming of coordinate measuring machines using NX

CMM Inspection programming is an application within NX system that may be used to determine complete, verifiable control programs for coordinate measuring machines (CMM) and NC (Numerical Control) digitally controlled machines with appropriate post processors. With the use of a configuration file of measure control that works in sync with the model file of NX part via CMM, Inspection Programming uniting the production sphere with production planning sphere as well as with design in such a way, sothat groups of technologists and engineers responsible for the construction could easily share data unambiguously related to inspection.

In the course of this project the students will learn how to:

- Use the CMM command and alignment tools in order to start the CMM inspection program and align the axes of parts to the axes of CMM machine by using the strategy of flattening of the surface /line/ /point/.
- Perform the assembly of an existing model of the measuring probe.
- Set the tracking points on the probe and the directions of appropriate vectors.
- Export the new measuring tool of the probe to NX library.
- Configure the machine head to work with a replaceable probe.

- Define how to connect the node tool in such a way that it will be possible to load the probe out of the tool cabinet during the simulation.

Add new parts files to the database of CMM and how to check his/hers work.

1.9. Ways of defining information concerning products and manufacturing (PMI) in NX

Product and Manufacturing Information (PMI) is the NX environment used to enclose non-geometric information, such as text, dimensions and symbols, to the part file. Creating PMI components can occur when the user is located in GATEWAY or Modeling module. Added information is later used by dependent applications, such as modules used for designing of tooling, manufacturing, controlling and transporting.

In most of the cases there is a possibility to attach information to any object in the part file.The information is displayed in 3D space which allows defining more useful information than it is possible on 2D drawing.

In the course of this project the students will learn how to:

- Use the PMI toolbar.
- Change preferences by editing PMI.
- Use the Part Navigator in order to edit PMI.
- Set the annotation plane.
- Organize PMI by related target objects.

Set preferences in order to determine the PMI objects' visibility.

1.10. Topology of structures optimization

Geometry optimization is commonly used in order to find the best solution in the context of the design goal, i.e. minimizing weight, peak deformation or displacement. The optimization process requires performing a set of simulations during which design variables are being adjusted based on the results of previous iterations. The process lasts until the design goal is achieved and the project meets all specified limitations.

In the course of this project the students will learn how to:

- Create FEM task, simulation file and solution for the model.
- Create a process of resolving optimization topology.
- Determine the FEM components attached to optimization.
- Determine geometrically restricted component that should not be modified during the optimization.
- Define the overall goal of optimization and respond what should be minimalized or maximized by the optimization project.
- Specify the attributes that are sensitive because of optimization changes and determine appropriate restrictions for these changes.
- Define the smoothing parameters of the model.
- Determine the control parameters, such as the number of iterations of the optimization.

- Perform the termination of the optimization model.
- Analyse optimization results.

Import the smoothed model to NX Modelling and use as a compound as a reference for creating a new project.

1.11. Defining of kinematic systems in NX

Motion Simulation module may be used to simulate motion and to evaluate mechanic systems in terms of:

- Displacements, velocity and acceleration.
- The range of motion.
- The force of reaction, the force of inertia and torque, and torques transmitted between segments.
- Interference of parts.
- Transferring the load obtained through kinematic simulation to modelling the processes with the use of the finite elements method.

It is possible in NX to use the assembly constraints for a motion simulation without the usage of Motion Simulation application. Still, the use of Motion Simulation module provides the following additional possibilities:

- Using a function to direct the motion of the model.
- Using the force and torques to direct the motion of the model.
- Evaluating the force and torques in the system.
- Defining the contact between the model and bodies or geometrical compounds, such as surves or points.
- Defining the characteristics of mechanical compounds such as springs and dampers in the model.
- The control of the mechanism of the model in an open loop and in a closed loop.
- Controlling the mechanism by the usage of the control chart prepared using the simulation tools in MALAB® Simulink®.
- Analyzing the deformation and stiff body motion in the susceptible elements.

In the course of this project the students will learn about:

- Types of simulation that may be performed using Motion Simulation.
- The basic structure of catalogues and files in part motion simulation.
- How to use basic tools, such as Motion Navigator and Motion toolbar.

1.12. Visualization of parts and styling assumptions

In the course of this project the information on how the user should use basic visualization tools will be presented.

This will include:

- Applying various effects to drawing in order to enhance details or simplify the image of the model.
- Providing information on how to control the lighting of the basic model.
- Using a camera in order to capture the characteristics of viewing the current workspace.
- Using cutting tools to show cross-sections of the model.
- Providing information on how to load photos and assign them to the walls of the model.
- Setting different types of rendering scenarios.
- Using raster images to assist in the design process.
- Setting exemplary textures of materials, lighting and bitmap reflections to create a visualization scenario possible to work in.
- Applying system scenes, editing them with the scene editor and creating own scenes.