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FROM I-DEAS TO NX

Changing the Design Application and Creating Work Instructions for a Large Industrial Product Assembly

Technology and Communication
2014

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Kone- ja tuotantotekniikan koulutusohjelma

ABSTRACT

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Title	From I-Deas to NX Changing the Design Application and Creating Work Instructions for a Large Industrial Product Assembly
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The objective of this thesis was to provide Citec Oy Ab with information whether it is beneficial to switch from using Siemens I-Deas to using Siemens NX, as the CAD design and drafting application, within a specific product group of Citec’s customer, an industrial product company.

To acquire this information a case project in NX was made. This was the main method of this project type thesis. The project work included building the products main top level assembly and creating an assembly drawing.

The results of the case project include the first NX assembly for this product group, an assembly drawing, and work instructions for future projects. Facts supporting the change and facts to be considered while switching were also listed, as well as a recommendation.

The recommendation provided by this thesis to Citec is clear: the switch to NX should be made. The remaining questions are when, as well as how to address the issues the switch raises.

Keywords NX, I-Deas, Teamcenter, Multi-CAD, interoperability

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TIIVISTELMÄ

Tekijä	Harri Mäkinen
Opinnäytetyön nimi	I-Deaksesta NX:ään Suunnittelutyökalun vaihtaminen ja työohjeiden tekeminen suurelle teolliselle tuotekokoonpanolle
Vuosi	2014
Kieli	Englanti
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Opinnäytetyön tavoitteena oli tarjota Citec Oy Ab:lle tietoa kannattaako sen vaihtaa Siemensin I-Deaksesta NX:ään asiakkaan erään tuoteryhmän CAD-suunnittelussa. Asiakas on valmistavan teollisuuden yritys.

Opinnäytetyön päämetodina tämän tiedon hankkimiseksi oli tehdä yksi projekti NX:llä. Työhön sisältyi tuotteen ylimmän tason kokoonpanon rakentaminen sekä kokoonpanopiirustuksen toteuttaminen.

Projektin tuloksia ovat ensimmäinen kyseisen tuoteryhmän NX-kokoonpano ja kokoonpanopiirustus. Lisäksi opinnäytetyöhön kuului laatia työohjeet tulevia projekteja varten. Yhteenvedon yhteyteen listattiin vaihtamista puoltavat seikat sekä vaihtamisen yhteydessä selvitettäviä asioita, sekä suositus vaihtamisesta.

Suositus on yksiselitteinen: Citecin tulisi vaihtaa NX:ään. Avoimeksi jäävä kysymys on milloin, sekä miten suhtautua vaihdoksen vaikutuksiin.

Avainsanat NX, I-Deas, Teamcenter, Multi-CAD, yhteistoiminta

TERMS AND ABBREVIATIONS

3D-CAD	Three-dimensional modelling
ASSEMBLY	A gathering of parts and subassemblies
BREP	Boundary Representation; a solid described by its outer boundaries
CAD	Computer-Aided design
EXTRACTED CURVES	Drafting lines separated from originating 3D
FACET BODY	A simplified type of a shell model
I-DEAS	A Siemens CAD application
IGES	Initial Graphics Exchange Specification; a neutral CAD file format
JT	A 3D data format developed by Siemens
LIGHTWEIGHT MODEL	An NX functionality for opening 3D in lightweight
MULTI-CAD	An environment with several design applications
NEUTRAL FILE FORMAT	A file format independent of application providers
NX	A Siemens CAD application
PARAMETRIC DESIGN	3D design which changes based on set parameters
PARASOLID	A file format for file transfer between CAD applications. Also the kernel of geometric design functionality in a CAD application, developed by Siemens.
PDM	Product Data Management

PLM	Product Lifecycle Management; a newer and wider term replacing PDM
STEP	Standard for the Exchange of Product Model Data; a neutral CAD file format described in standard ISO 10303
TEAMCENTER	A Siemens PLM application

TABLE OF CONTENTS

LIST OF FIGURES

1 INTRODUCTION

In industrial design, it is not an easy and everyday task to switch from one design solution to another. Mechanical design is these days almost exclusively done with 3D-CAD applications (Computer-Aided Design). These applications are costly and complicated. They are often used in connection with PLM applications (Product Life-Cycle Management), which are even more costly and complicated.

A large industrial company often uses one CAD/PLM solution for a decade or even decades. When switching, the solution typically comes from the same solution provider, to allow for a reasonable amount of effort; switching to a different providers CAD/PLM environment may - in the worst-case - require a complete redesign of all products.

When a decision to switch is made, there are many viewpoints to consider: /17/

- Arguments for and against the switch
- Capabilities of the considered CAD application(s)
- What amount of re-design is needed
- Compatibility with other used internal and external CAD systems and file formats
- Software and hardware requirements and costs
- Upgradability and expandability
- Support availability and cost
- Existing product knowledge; building the missing competences
- Within engineering services: customer requirements

This thesis aims to look into a switch from Siemens I-Deas, originally developed in the 80's, to the more modern Siemens NX. There will be a case project, in which a very large assembly of a complex machine will be built and the necessary mechanical drawings will be created, using NX.

The case provides a relatively easy situation for the switch: both the CAD application to be replaced, as well as the one replacing it, are by the same software solution provider. Furthermore, they both interoperate with the same Product Life-Cycle Management (PLM) solution, Teamcenter, and can through a solution called “Multi-CAD”, to a degree, use the one and same product data and designs.

Complicating the switch though is the fact that for an intermediate period of time both I-Deas and NX will need to be used simultaneously. They will need to interoperate and be able to exchange product data and designs, although only in one direction: from the older I-Deas to NX. In the case project, options will be tested for the data exchange between I-Deas and NX.

Furthermore, Citec as well as the customer operate in an environment of multiple CAD systems. Such a “Multi-CAD” environment requires that product data and designs can be exchanged between solutions of up to 10-15 applications. Internally (at Citec and the customer, since Citec operates within the customer’s internal network and tools), there are several disciplines involved with favoured applications: civil, electrical, mechanical and process. Externally, Citec and the customer vendors provide designs with a multitude of systems.

In the case project some of these external formats will be tested to see how they work for the purpose of the case project.

The case project will be done as a service to a multi-national industrial customer of Citec Oy Ab, an engineering services company. These services have been provided with Siemens I-Deas since 2004.

The industrial customer and its products will not be identified in this public thesis report.

1.1 Thesis Objectives

The objectives of the thesis are to:

1. Research whether it is possible to use Siemens NX instead of the older I-Deas in the design and 3D assembly models of one product in one specific customer product group and to make the related drawings.

2. Create internal work instructions to be used within these services
3. Research and list arguments for and against the change and to provide Citec Oy with a recommendation whether to change or not

1.2 Methodology

The thesis is based on a case project. Scientific research methods have not been used.

Due to strict confidentiality requirements from the customer this public thesis report only contains a broad overview of the case project and no detailed information of the results.

2 PARTNERS

The case project was done as a service provided by Citec Oy Ab to an industrial customer.

2.1 Citec Oy Ab

Citec provides multi-discipline engineering and information management services to technology-dependent industries. The company aims to “secure our customers’ success with high performance, expertise and passion”. /4/

Furthermore, the company describes itself as “striving towards improving the value and usability of your products and processes in multi-discipline engineering and technical communication. Our offshore solutions, in combination with our local customer service, provide our customers with clear advantages in terms of competence, cost efficiency and high quality. Thanks to our solid global experience, we are able to achieve the results needed.”

The company was founded in 1984 by two engineers, who met during their engineering studies in Vaasa, Finland. The company started to grow rapidly in 1989 and has continued to do so ever since. The number of Citec employees currently amounts to approximately 1100 and the turnover of 2013 was approximately 70 million euros. Citec is headquartered in Vaasa, Finland, and has offices in Finland, Sweden, Norway, UK, France, Germany, Russia, India, and Singapore.

For the first decade of 2000’s Citec was split into three separate companies: Citec Engineering, Citec Information and Citec Environmental. The companies were united when a fund managed by Sentica Oy bought the majority of the company stocks in 2012. Until then the founders and company management owned the company.

Citec provides engineering and technical information services for these main sectors of the company:

- Process Industry and Manufacturing
- Civil
- Vehicles
- Healthcare
- Information Communication & Technology (ICT)
- Energy & Power
- Oil & Gas

Figure 1. Citec Headquarters in Vaasa.

2.2 Customer

The customer of this project wishes not to be identified and considers the services Citec provides to it fully confidential. Therefore, the customer cannot be identified in this public thesis report.

The customer is a multinational industrial product and services company. Its annual turnover is in the billions. It manufactures heavy industrial products, as well as provides related project services and operation and maintenance services.

3 CASE PROJECT BACKGROUND AND REQUIREMENTS

3.1 Current Status at Citec and the Customer

Citec provides a multitude of services to the customer, including participation in the design of one of their main product groups. Citec also currently creates the main top level assembly drawing for each customer product within this product group.

3.2 Current Design Application: I-Deas

Within the services, many design applications are used. For the product group which includes the product of the case project, the currently used design application is Siemens I-Deas. I-Deas was originally developed by Structural Dynamics Research Corporation in 1982. After several ownership arrangements, it became a product of Siemens AG in May 2007, within their Siemens PLM Software product group. /10/ /11/

I-Deas is a powerful 3D CAD application and it is used by many large industrial companies, including the globally operating Ford and General Motors. Nevertheless, its user interface is outdated and it is increasingly difficult for companies to find personnel trained to use it. Also, its functionalities are outdated: for instance its parametric modelling capabilities are limited. As a result, Siemens is strongly directing its customers to abandon I-Deas and move to their newer offering, NX. /20/

The official Siemens product name for I-Deas is “NX I-Deas”. To separate it from the newer NX application, in this thesis report the name is shortened to “I-Deas”, which is also the common spoken name of the product.

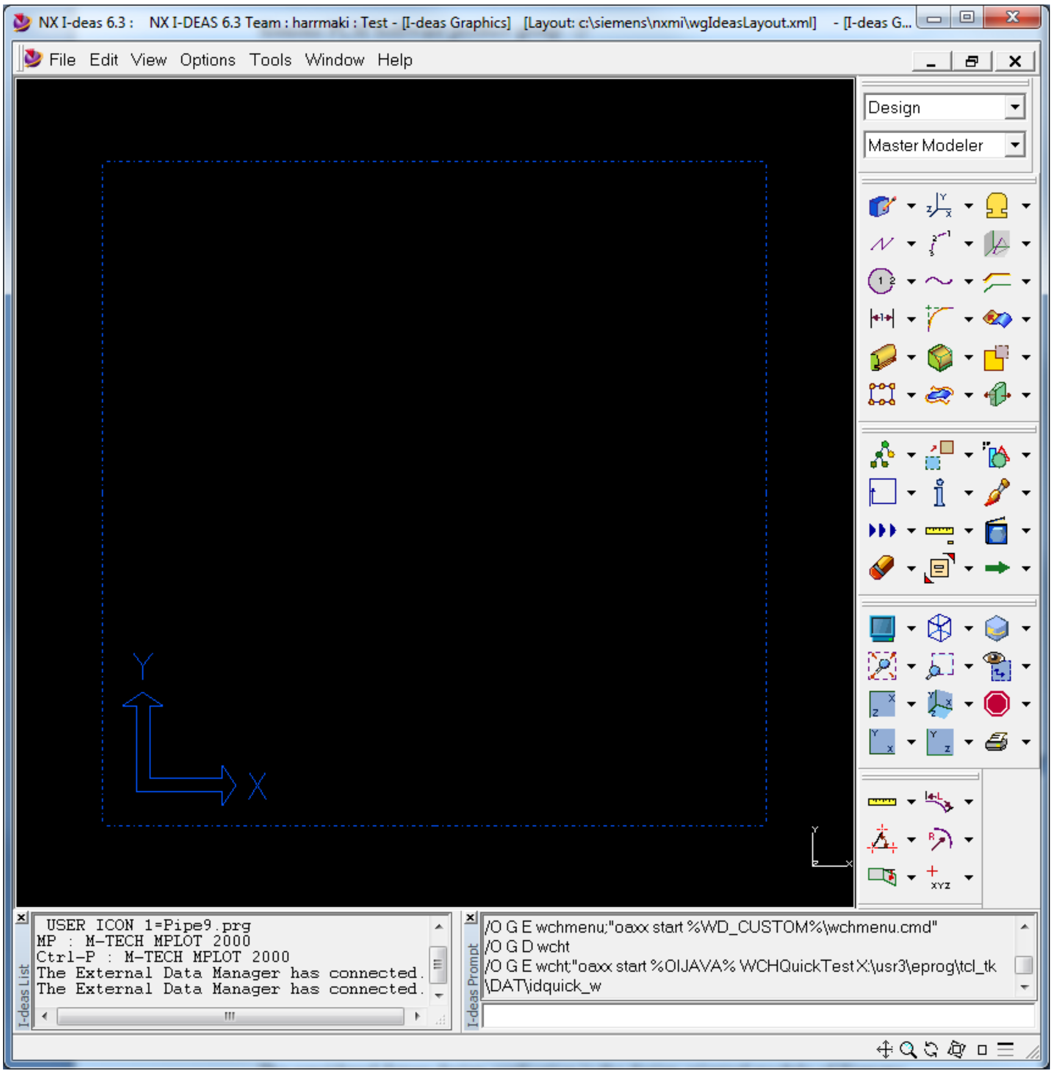

Figure 2. The I-Deas 6.3 interface.

3.3 Considered Future Design Application: NX

The considered future design application is the design oriented module of Siemens NX, “NX for Design”, from now on “NX” for short. NX, in contrast with I-Deas, has a modern user interface, which functions as expected by users familiar with modern applications designed to run in the Microsoft Windows operating environment. /1/

NX is modular and includes modules for computer-aided design (CAD, manufacturing (CAM) and engineering analysis (CAE). Citec uses it to provide services to several customers.

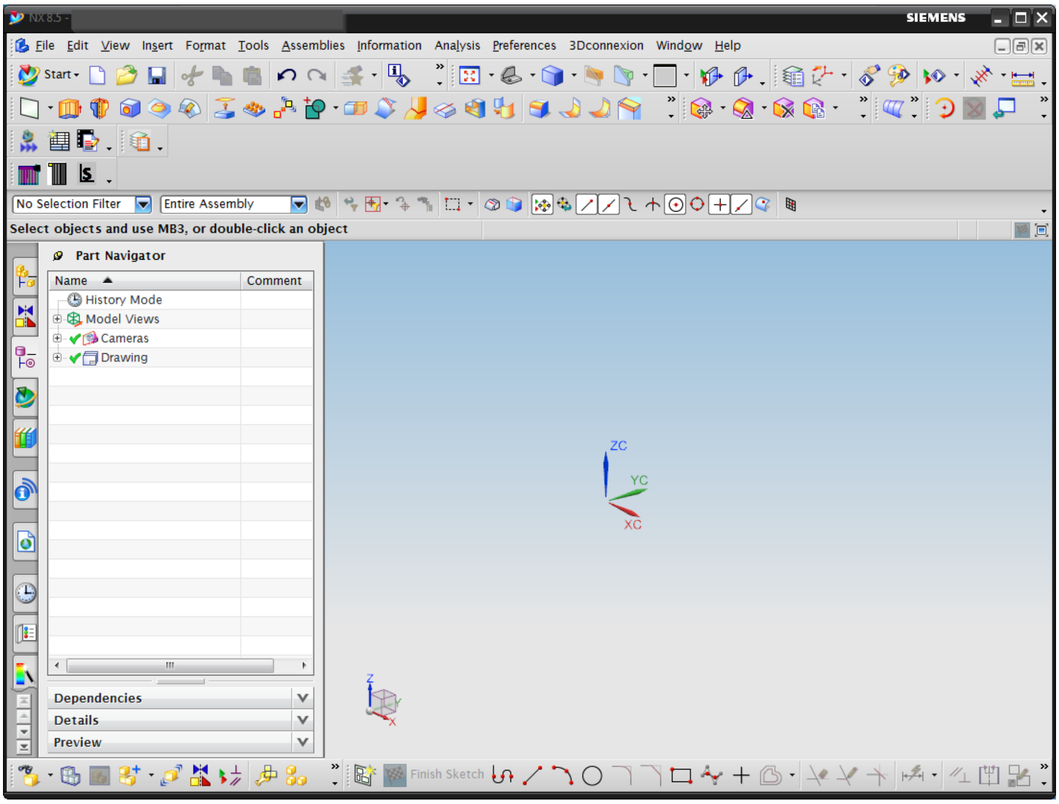

Figure 3. The NX 8.5 interface.

3.4 The Switch from I-Deas to NX

Citec's customer has a project which aims to migrate all product design from I-Deas to NX. The customer-specific environment of the related PLM application, Siemens Teamcenter, is already developed mostly with NX in mind; with each environment upgrade, working with I-Deas is increasingly more difficult. This is one of the main arguments to move the relevant design to NX at Citec also. /18/

The schedule of the customer switch is undecided. New products are systematically designed in NX; the designs of products further in their life-cycle are mostly kept updated in I-Deas. The products are mass-tailored, which means that the designs are not frozen but continue to actively change. The customer plans to decide during 2014 how to make the switch to NX within these older products.

The key arguments Siemens uses in promoting switching from I-Deas to NX are: /21/

- NX is a modern design application with a modern user interface
- Lesser learning curve
- I-Deas development and support will eventually end (according to current information in 2015)
- NX has a wider scope than I-Deas (see figure 4)
- Parametric modelling capabilities are better in NX
- Modern part and history trees
- Tight integration with Teamcenter
- Development and adoption of new technologies

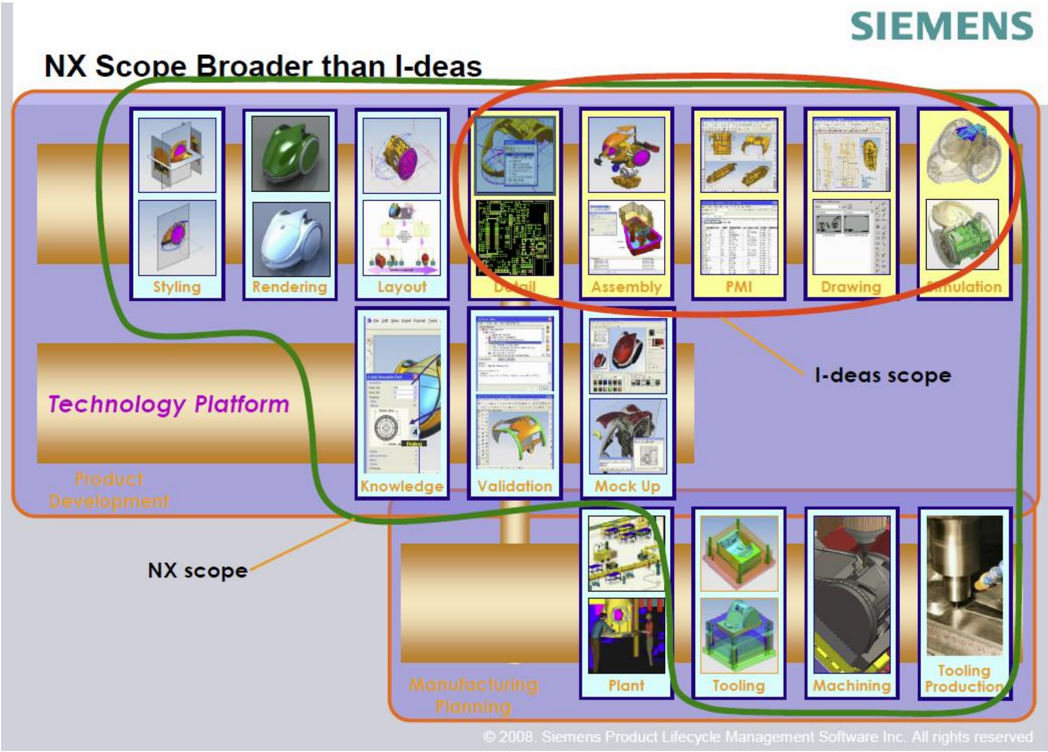

Figure 4. NX and I-Deas scope comparison. /7/

4 INTEROPERABILITY WITHIN CAD SYSTEMS

As described above, Citec and the customer are in a situation where I-Deas and NX will need to be used simultaneously for a period of time. Furthermore, it is necessary to be able to exchange data with other CAD solutions, within Citec and the customer and within the network of vendors and the products' value chain.

This exchange of data is typically called the "interoperability" of CAD applications. It can be achieved by four methods: /3/

1. Direct import and export in CAD systems
2. PLM system support for the native data formats of competing CAD systems
3. File format translators
4. Intermediate or "neutral" file formats

An environment which needs to support multiple CAD formats is often called a "Multi-CAD" environment. Siemens uses this term to describe their support for their own older CAD formats and competitor formats.

4.1 Direct Support for External CAD Formats

Modern CAD applications support tens of file formats. The Siemens NX can directly import designs from several applications, including these widely adopted ones: AutoCAD, Pro/ENGINEER and CATIA. It can also import and export many neutral file formats, such as IGES, STEP, Parasolid and JT. /5/

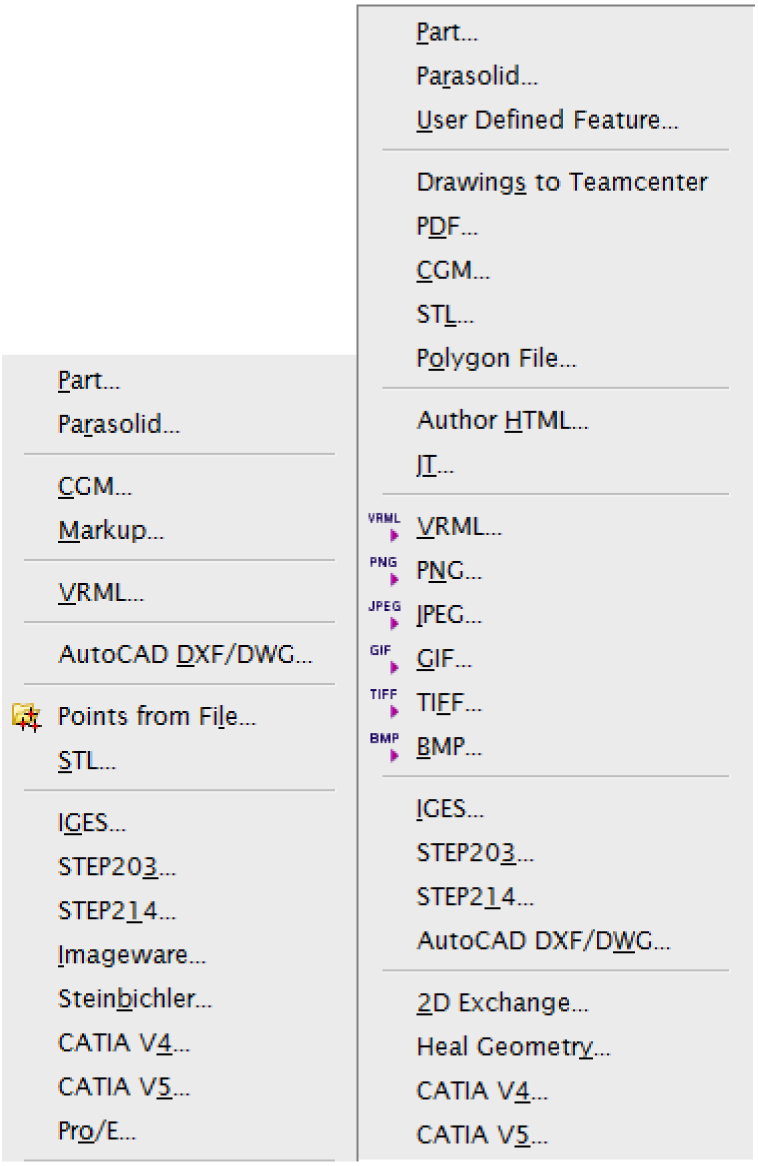

Figure 5. NX import and export options and support.

Another approach for providing interoperability is within the PLM environment. This approach aims to include data storage for product data and designs from another application and provide these designs to NX as if they were in its own native file format. These solutions are modularized and not available in all Teamcenter installations.

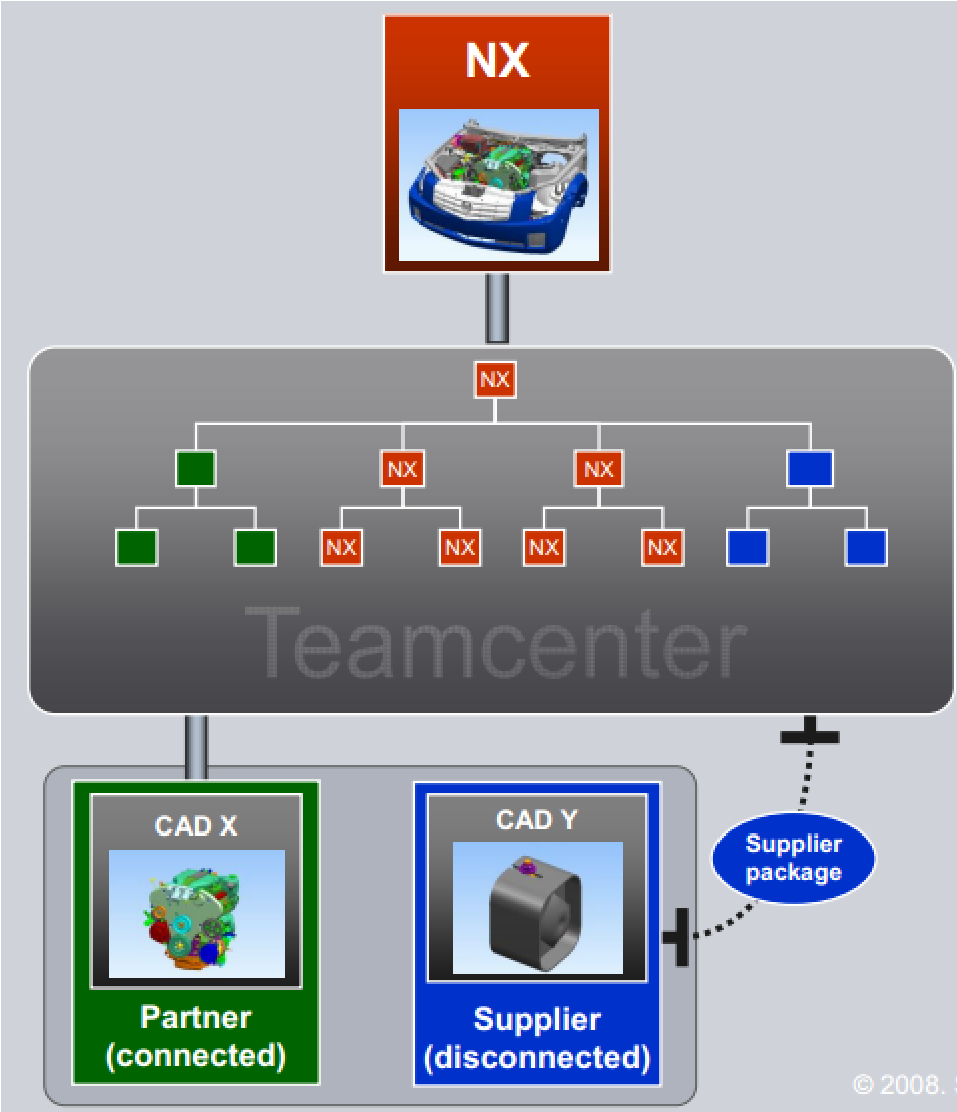

Figure 6. NX can reuse designs from other CAD applications. /22/

4.2 Third Party Translators

There is a multitude of third party translators of CAD data available. These are outside the scope of this thesis.

4.3 Neutral File Formats

To tackle the interoperability problem, several neutral file formats have been created. These are standards based formats which are typically supported by all CAD applications. Within mechanical engineering, the most common neutral file formats are IGES and STEP.

The use of neutral file formats introduces organizations to major drawbacks:

1. Product intelligence cannot be transported with them, such as material numbers, metadata and relations. Traceability is lost.
2. They cannot transfer the design history
3. Revision control is difficult or even impossible
4. The imported geometry may be difficult or impossible to edit further
5. They add to the risk of design geometry errors

4.3.1 IGES

IGES, the Initial Graphics Exchange Specification, was originally developed in the 80's, but it is still widely used, although STEP is increasingly the more used neutral file format. The file name extension of an IGES file is *.igs*. /3/

4.3.2 STEP

A STEP file, Standard for the Exchange of Product Model Data, is often seen as the successor of IGES. It is based on the ISO 10303 standard and currently commonly used as versions AP203 and AP214. The AP203 version defines the geometry and topology of mechanical parts and assemblies. AP214 adds colours, layers, geometric dimensioning and tolerances, as well as design intent. A near-future new version AP242 will further develop the STEP format. The file name extension of a STEP file is *.stp* or *.step*. /12/ /16/

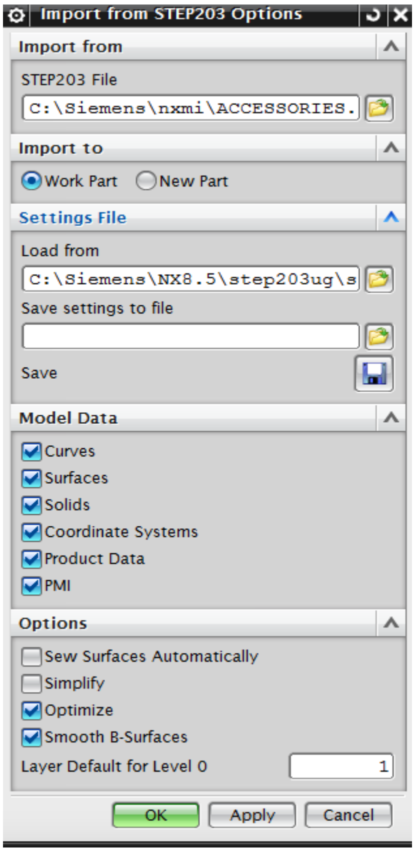

Figure 7. STEP import to NX.

4.3.3 JT

JT is a 3D data format developed by Siemens PLM software but has since become an ISO standard (ISO14306). It can be seen as a neutral file format although it is commonly used for transferring assembly structures for visualization and presentation purposes, instead of part geometry, while it is capable for this purpose also. The file name extension of a JT file is *.jt*. A free viewer of JT models is available. /15/ /16/

4.3.4 Parasolid

Parasolid is the geometric modelling kernel of I-Deas and NX, developed by Siemens, but it can be licensed by other application developers to be used in their products. It has been widely adopted for this purpose.

In addition, Parasolid is a file format, which can be handled with any application using the Parasolid modelling kernel or which includes a translator for it. Its file ending is either *.x_t* or *.x_b*, where *t* stands for text format and *b* for binary format. /19/

4.4 Popularity of the File Formats

According to a 2013 survey, among 844 companies (world-wide but with emphasis in the US) utilizing CAD, STEP was by far the most popular exchange format, with 79 percent of respondents using it. The older IGES was used by 58 percent of the companies, 3D PDF by 41, Parasolid by 36 and JT by 22 percent. /13/

4.5 I-Deas Designs via Siemens Multi-CAD to NX

In the case project, the designs exist in the native I-Deas format and are stored within the Teamcenter PLM environment. Siemens has provided a Multi-CAD module, which was originally implemented in 2012 to the customer environment, to allow the use of I-Deas designs in NX.

In this solution, the designs are translated via JT packages, including Parasolid geometry, as boundary representations (BREP) and called XT-Brep. The XT-Brep data does not exist within the customer Teamcenter PLM environment prior to 2012, and it appears to be typically only usable in NX if created after the beginning of 2013. The newer, the better it appears to be. In a case of the data missing or incomplete, the agreed way-of-working is to create a request to the customers IT helpdesk, asking for creation, correction, completion or update of the XT-Brep data. /14/

4.6 The Challenges of a Multi-CAD Environment

An environment with multiple CAD solutions or even file formats can become problematic for a company. This was clearly proven to be the case in the customer environment, as described later in chapter “Case Project Overview”.

Waldenmeyer and Hartman have made a case-study of a Multi-CAD implementation in an academic environment. According to their article, an environment with multiple competing CAD applications can “become faced with high licensing and training costs, compounded by the fact that competing CAD packages do not communicate well, if at all”. /22/

As a solution, Waldenmeyer and Hartman go as far as to suggest that the use of neutral file formats should only be considered an exception, not to be systematically used in a product design environment. Instead, they promote the development of PDM/PLM systems, such as the Siemens Teamcenter, to include abilities to retain product data and design history, a “Multi-CAD environment”.

A commercial research study made by Michelle Boucher for the Aberdeen Group, a US consultation company, suggests the neutral file formats are “dumb blocks”. According to Boucher, a neutral file provides the general shape of a part, but “not much can be done with it. Typically, changes to the geometry are limited, if even possible.”. /2/

According to Boucher, “it is obvious that organizations are moving away from using neutral files such as IGES or STEP for file translation, but instead relying on direct translation of native CAD files into third party CAD”. The translation of native CAD is then typically done through PDM/PLM systems.

This was clearly the case in the case project. Therefore, the use of neutral formats was limited to cases where it was impossible to transfer the data through the Siemens Multi-CAD solution, due to limitations of the solution.

4.7 Limitations of the Siemens Multi-CAD Solution

The Multi-CAD solution saves a great deal of cost for the customers of Siemens. Unfortunately, whenever Multi-CAD assemblies or parts are used, also major design limitations are introduced:

1. Multi-CAD excludes the use of the lightweight representations, which is a major disadvantage when working with very large assemblies.

2. Multi-CAD excludes the use of the Extracted Curves functionality, which limits the possibilities of solving the problems such large assemblies create.
3. Similarly, any existing Facet Bodies cannot be used
4. Multi-CAD data only exists in new designs, created after the introduction of the Siemens solution. In the customer case, this was 2012. Earlier designs require a helpdesk ticket or a revision, to create the XT-Brep data.

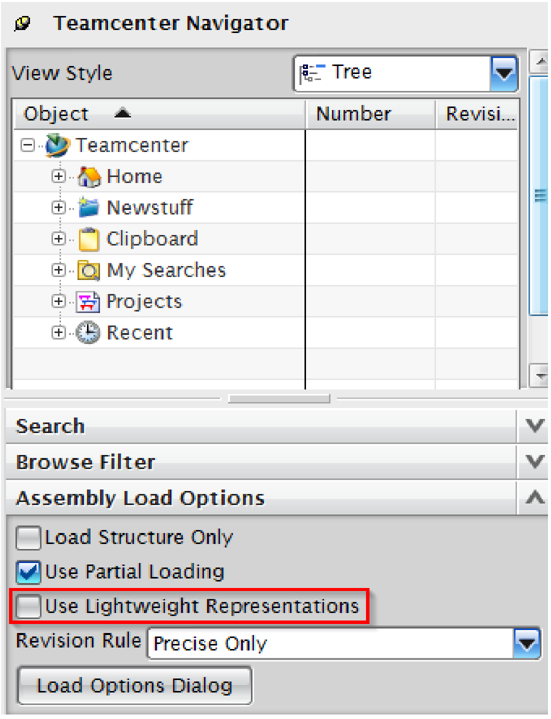

Figure 8. Lightweight Representations cannot be used.

4.8 Quality of Translated Data

The quality of the translated Multi-CAD data was mostly acceptable for the purpose of this project, as the translated data was not used for design, but instead only for viewing and building a new assembly.

For design purposes, translated data can be less useful. At Citec, designers typically redesign imported designs in NX. They say a design “just works better in the native format”. As reference and background in assemblies they find translated non-native 3D usable. /8/

In a CAD evaluation project for CERN (European Laboratory for Particle Physics), Friman and Wikner found these characteristics usable in evaluating the usability or translated 3D data (listed in their increasing order of importance): /6/

1. Colours OK
2. Enough faces to view part of solid
3. Solid is without any missing shape (e.g. cut-out or fillet not represented)
4. OK for viewing, some missing faces
5. Time to repair into originally shaped solid is less than 30 minutes
6. Volume OK
7. Number of faces OK
8. Centre of gravity OK
9. Possibility to modify solid

For purposes of design, it appears item 9 of the above list is not yet working adequately in the customer environment. It is possible though that the quality level of the translation could be raised by researching further what specific design features are causing the problems. According to Tornincasa and Di Monaco, for instance fillets and chamfers should not have children. Such problematic features could be found by design checking tools integrated into many CAD tools as well as available separately; they typically check for things like sliver faces, cracks, internal voids, self-intersecting curves and surfaces, degenerated entities and overlapping edges. /20/

5 CASE PROJECT OVERVIEW

This description in the public thesis report is an overview of the key phases and findings of the project.

5.1 Objective

The objective of the case project was to create the highest level full assembly and the assembly drawing in NX for one of the customers products. For this product group, this work has been done with I-Deas for the past 10 years. During this time, the work has been outsourced to Citec.

The top level assembly drawing has several purposes:

- It is used as input by other design departments of the customer
- It includes the main dimensions, weights and Centre of Gravity of the product, for civil and structural design purposes, as well as transport and lifting
- It includes process related information, used as input by process design
- It is delivered to the product user, within product documentation

5.2 Part Design

NX is already used in part design and for building the sub-assemblies for newer customer products, as well as by other product groups. Therefore, it is clear that NX could be used in part design for this product group also.

But for the case of this product group, most current designs exist in I-Deas format. I-Deas part designs have successfully been used in part design in NX, through the Siemens Multi-CAD solution. The solution is flawed or not fully developed yet though: the designs are mostly usable as reference or background only, not for continued design.

With this prior knowledge that part design is possible through Multi-CAD, the open question remains if the top-level product assemblies can be built with NX.

5.3 Schedule

According to the schedule of the thesis work, the case project was planned to be made during December-January. One project was first chosen but abandoned due to a tight customer schedule. Another project was then chosen. This projects delivery of the assembly drawing was required by late January 2014, which suited the thesis schedule perfectly.

Necessary inputs, including the customer’s internal product order and the top level product structure, were available sometime before the years end.

It was possible to make most of the work during late December - January and deliver a preliminary drawing by the end of January. The final drawing was then approved February 10. This was acceptable by all users of the drawing, including the customer.

This schedule was extremely tight. Abandoning NX and going back to I-Deas was very close twice, as it looked like it would not be possible to deliver the project in time. This was due to two factors:

1. The data size of the assembly became very large. It became very slow to open to NX.
2. The desktop computer in use, although a high-end 3D design workstation, could not handle the assembly. Arranging necessary hardware upgrades took time.

It was possible to solve these issues, to some degree, as described later.

5.4 The Product

The product of the case project is a large-scale industrial product, with approximately 90 sub-assemblies (or product modules) on the second level of the product structure. In all, the product has some 8000-12000 individual parts. With the exchange of the product modules, the product is mass-tailored for each customer project, according to a project specific configuration.

5.5 Limitations

From the start, there were key limitations to what NX functions and what way-of-working could be used.

Almost all of the product modules were I-Deas designs, with only one module available as an NX design. This caused several complications. First, due to the features of the Siemens Multi-CAD implementation, otherwise usable NX functionality could not be used, as described above in chapter 4.6 “Limitations of the Siemens Multi-CAD solution”. This was an issue because the unavailable functionality could have been used to reduce the problematic size of the assembly.

Secondly, for many older I-Deas designs, the Multi-CAD 3D data was missing, not up-to-date or otherwise flawed in Teamcenter. In such a situation, the agreed way-of-working was to create an IT helpdesk ticket, requesting to create, update or fix the data. This caused extra communication and added to the duration of the project.

These limitations and drawbacks were a major challenge to creating the assembly drawing in NX and should, if possible, be addressed in future updates of the PLM/CAD environment.

5.6 Building the Assembly

The product is designed around a coordinate system, whose zero point is called “the Basic Point”. This point is located at the same position in each product of the case projects product group, at a certain location of a main product part. The full assembly was built around this part and the Basic Point. This took several weeks.

The full assembly could not be built with the subassemblies as they were, since the end-result would be too large for any computer to handle. Instead, visible or otherwise necessary parts had to be copied to the top assembly level, by first bringing in the full subassembly, then copying necessary parts and last deleting the subassembly.

Most subassemblies were with reference background, which is not included in the assembly's part list (BOM, Bill of Materials). Very often the background included a key part which could be used as a convenient way to position the components.

5.7 Constraints

The customer does not use constraints in building the top level assemblies. To see if using constraints would provide benefits, they were used to position some of the main parts. But in the end, use of constraints was not beneficial. Also, senior designers advised against the use of constraints. With an assembly of this size, according to them, the result would be a "constraint hell", specifically during and after years of revisions by different designers. /8/

5.8 Use of Neutral File Formats

Some older designs refused to work through the Multi-CAD solution; the customer's helpdesk could not fix them, and it was not acceptable to revise them for this reason only. In such situations, the designs were transferred through Parasolid or STEP formats.

Typically Parasolid, being a Siemens solution and file format, provided the best results: all features worked and for instance holes could always be used in positioning parts. With STEP, even if exported with "all features" from I-Deas and imported similarly to NX, features were sometimes missing and as a result positioning was difficult.

Generally the use of external file transfer was avoided, as was expectable from experiences of others (see chapter 4). Also, at some point all the I-Deas designs will need to be made to work in NX. With this aim in mind, the established way-of-working was used e.g. helpdesk tickets were created whenever Multi-CAD data was missing or not working. The external formats were only used when it was the very last working solution. Another important reason for this was to not loose upgradability to newer revisions and traceability.

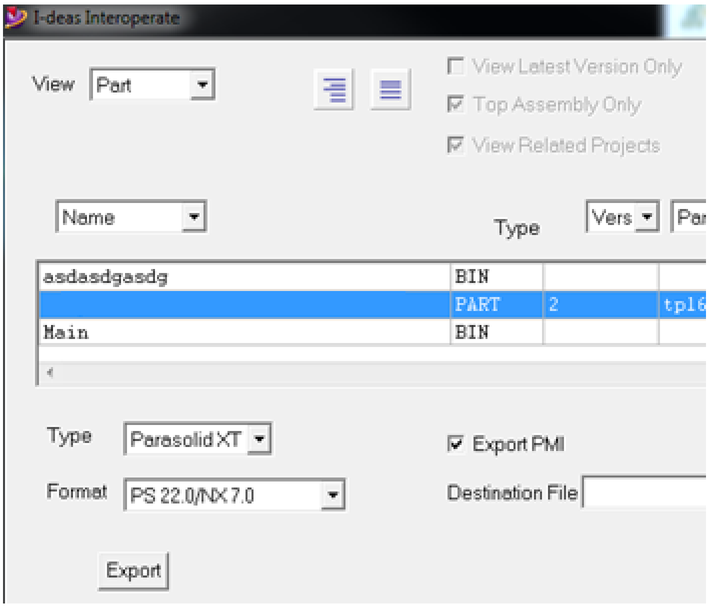

Figure 9. Exporting a part to Parasolid format from I-Deas.

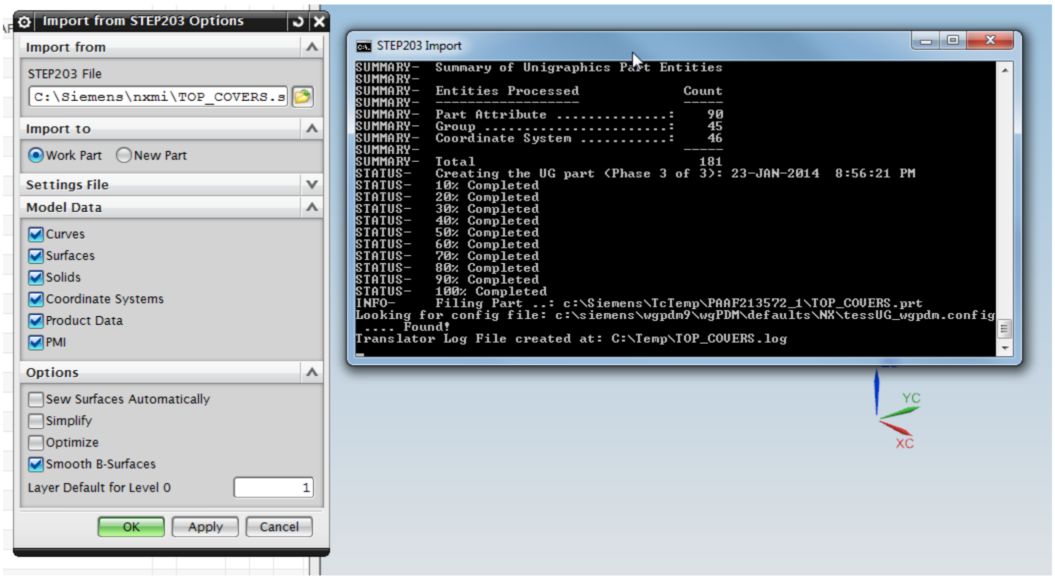

Figure 10. Importing a STEP file to NX.

5.9 Assembly Size and Solutions

With approximately one third of the model built, the design workstation started to get very slow. Also opening the model started to take 35-45 minutes each time, including after each crash, which there tended to be once or twice a day. It turned out NX does not handle very large assemblies very well. A similar size assembly handles without problems in I-Deas.

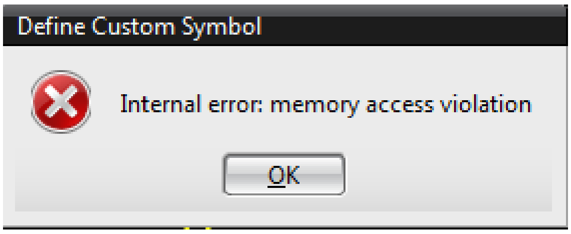

Figure 11. The large assembly made NX unstable.

Three solutions to this problem were found:

1. Hardware upgrade. The used workstation was already high-end, but still the graphics card had to be replaced and the RAM memory amount doubled. It was learned that for this work in NX, a very high-end workstation is needed. For exact specifications, see chapter “Recommendation” below.
2. Splitting the assembly to three subassemblies.
3. Making the assembly essentially a shell, e.g. systematically removing all components which were not visible or necessary for positioning.

   While removing these materials, it was necessary to consider the needs of possible revisions and future projects which would be started from this assembly. Non-visible materials may be needed for these purposes, to not make future work too difficult or time-consuming. Also, parts were not removed from any Citec designs, to allow easy replacing.

5.10 Drafting

The Siemens Multi-CAD solution required a specific set of settings to be found, to achieve a clean and readable assembly drawing. With default settings, the drawing was an unreadable mess, with hundreds of randomly located lines and elements.

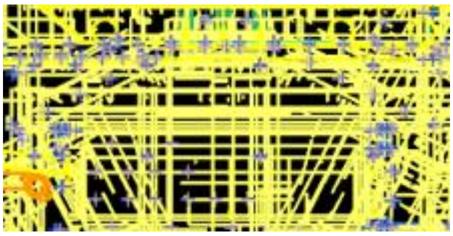

Figure 12. Part of the drawing with NX default settings.

The first tried solution was to hide the strange lines one by one. With more subassemblies added, this proved not to be a solution. Also, after each reopening of the model, the drawing became unreadable again.

Experiments with view settings were then made, as well as trying different assembly load and saving options. It was also necessary to systematically make sure each I-Deas assembly or part had up-to-date Multi-CAD data (XT-Brep).

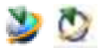

Figure 13. Teamcenter icons for valid and invalid Multi-CAD data.

The correct drafting settings and solutions found were:

1. The Multi-CAD data must be systematically up-to-date for each subassembly or part
2. Facet bodies and curves must be hidden with the "Show and Hide" function
3. In view settings, "Extracted Curves" must be unchecked and "Silhouettes" checked
4. The model must be opened with Load Option "As saved" (see figure 16)
5. The model must be saved with the "Save Precise Structure" menu option

With these settings and NX functions, the drawing became readable and neat.

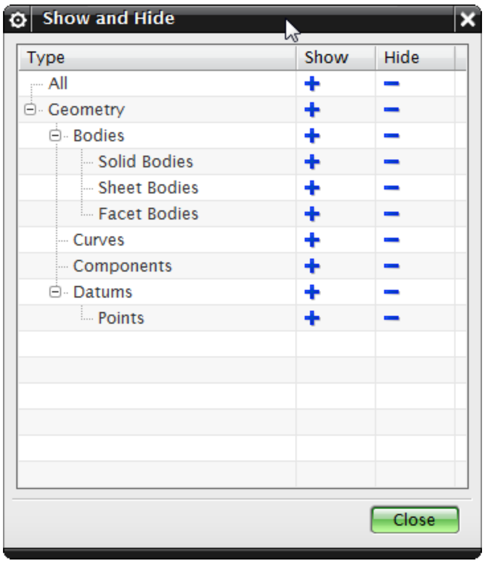

Figure 14. Facet Bodies and Curves had to be hidden.

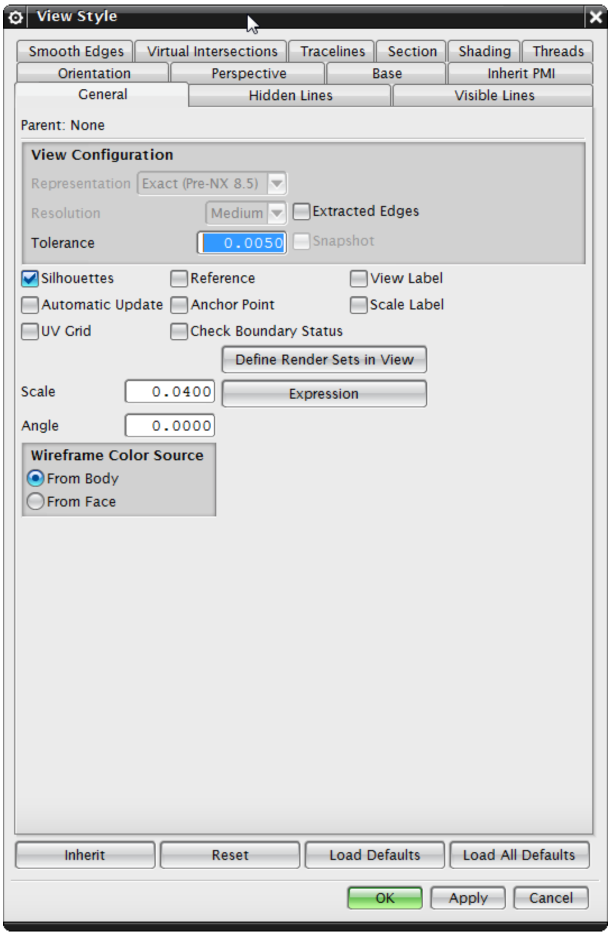

Figure 15. Usable view settings.

5.11 Problems from the Found Drafting Solutions

Hiding facet bodies created a problem though: many materials used in I-Deas were facet bodies. The solution was to remodel them in I-Deas and export the result trough Parasolid to NX. This is not an ideal solution as it takes unnecessary time and upgradability and traceability are lost.

Similarly, hiding curves removed the possibility of using the “Extracted Curves” functionality. It could have been useful in solving the data size and the opening time problem. Possible solutions for this exist (see chapter 7.4 “Further Development Options”).

5.12 Working Settings for Opening the Model

With I-Deas, the model must be downloaded only once from Teamcenter, when first starting the work. It can also be saved to Teamcenter only once, when finishing the work.

To allow simultaneous engineering, Siemens has moved away from this solution. In NX, each time the model is opened, it is loaded from Teamcenter. With default options, each sub-assembly or part is then automatically updated to newest approved revisions. Furthermore, each time the model is saved, it is saved to Teamcenter.

For the case projects assemblies, this is an undesired functionality, as long as most designs are in I-Deas format. A subassembly can include parts with corrupt or missing XT-Brep data, each potentially rendering the assembly drawing unreadable. The solution is to open and save the drawing and model with specific functions and options (see chapter 5.10 and Figure 16).

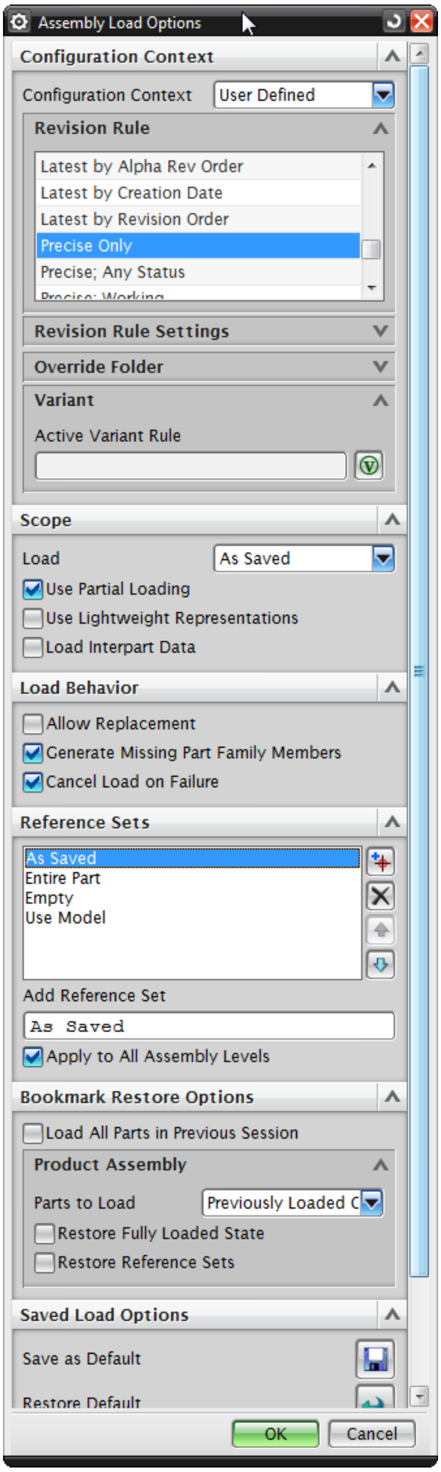

Figure 16. Working options for opening the model.

6 RESULTS

The results of the case project are:

1. The assembly model: first assembly built in NX for this product group
2. The main assembly drawing
3. Work instructions for creating top-level assemblies in NX for this product group
4. Facts for and against moving to NX and a recommendation

6.1 The Assembly model

The model is an assembly which includes all the mechanical parts needed for the purposes described in chapter 2. It is saved as “help material” in Teamcenter, e.g. it is not official “material”, which could for instance be used for purchasing purposes.

6.2 The Assembly Drawing

The product assembly drawing was created for a specific customer delivery project and product configuration.

6.3 Work Instructions

The work instructions are a detailed set of instructions for designers, which were made in MS PowerPoint format. They will be stored in Citec’s document management system, as well as delivered to customer upon request.

6.4 Recommendation

The final result is a recommendation which includes facts for and against moving from I-Deas to NX (see chapter ‘Conclusions’ below).

6.5 Use of the Results

The model and drawing will be used by one project as described earlier in chapter 2. They can also be used for further projects with exactly the same or a closely resembling product configuration. They will also be used as the reference for any future NX assemblies and drawings in the same product group.

The work instructions will be used by Citec designers, as well as potentially the customer.

The recommendation (switch to NX or not) will be used by Citec management as input to decision-making.

7 CONCLUSIONS AND RECOMMENDATIONS

Working in NX to build the model was very time-consuming, much more so than with I-Deas. For some reason, NX could not handle an assembly of this size, whereas I-Deas easily can.

On the other hand, Siemens does not develop I-Deas anymore, and in the near-future will stop supporting I-Deas. The customers Teamcenter environment is developed with mostly NX in mind, I-Deas seeming to be an afterthought. Working in I-Deas in the customer environment gets more and more difficult with each environment update.

During this project, I have learned that it is clear within Citec and within the customer that the switch to NX must be made. The customer already has a running NX implementation project, which includes abandoning I-Deas. This was not previously known at Citec. /18/

Therefore, it is clear that Citec must switch to NX within the services to this division of the customer organization and within the product group in question. The remaining questions are when and how. This thesis provides answers to "how".

In the changed situation, facts against the change are not listed here, which was an objective of this thesis. Instead facts for the change will be listed, to provide them to decision-makers in case of change resistance, as well as issues which will require attention and solutions.

7.1 For the Change

These facts support the switch:

1. NX is the future application which will replace I-Deas eventually and in customer use within the near future, likely within a period of up to three years. This schedule may exclude isolated areas which may keep using a standalone I-Deas.

2. NX has a larger scope than I-Deas (see figure 4), allowing the customer to benefit from using one data source for many purposes, and giving Citec possibilities to offer the customer services with a larger scope.

3. NX allows better co-operation with other departments and disciplines at Citec and the customer organization:

   - NX supports import/export of more CAD formats than I-Deas

   - It may become possible to provide lightweight shell models of the full product assembly to other uses within Citec and the customer. This would be a major advantage to Citec and the customer.

   - Simultaneous engineering becomes easier

4. Parametric design options become better. This is a future focus area in the customers NX implementation project.

5. NX has a modern user interface and it is easier to learn than I-Deas. Availability of personnel trained in NX is better.

6. With each Teamcenter environment upgrade, new problems with the Teamcenter-I-Deas integration tend to appear. Teamcenter appears to be developed mostly to work with NX.

7.2 Issues to Consider

These facts should be considered while planning and implementing the switch:

1. Top-level assemblies for other products within the same product group have not been built. Availability of Multi-CAD data for these products is unclear.

2. Hardware requirements are high and require a minimum:

   - Computer: HP Z420 or similar

   - Processor: 2 Intel Xeon E5-1620V2 3.7 GHz or similar

- Memory: A minimum of 24 GB
- Graphics: an extremely high-end graphics processor: nVidia Quadro K4000 with 3GT GDDR5 SDRAM memory or similar
- Hard drive: preferably an SSD type system HDD, with the system and applications fully installed on this drive.
- It is important to note that lesser hardware than listed will simply not work: the computer will not be able to handle the load.
- Also, the remote desktop solution used by some design departments cannot be built into the above configuration, mainly due to the GPU requirements. Another remote desktop solution would need to be built or local desktops would need to be used.

3. The opening time of the full assembly (optimize as described above) is 15-25 minutes, each time it is opened.
4. Building each product configuration within the product group and the main assembly and the drawing for the first time will require approx. a full personnel month or more.
5. Multi-CAD parts are suitable to be used as reference/background in NX. For design, they do not work as well as native NX parts. Typically, currently, Multi-CAD parts need to be redesigned in NX.
6. Training, to build the necessary NX competences.

7.3 Part Design

Part design was outside the scope of this thesis. It is relevant to remind though that in part design, it is not recommendable to switch to NX as long as the top-level assemblies and the main assembly drawing are made in I-Deas.

The Siemens Multi-CAD solution does not work from NX to I-Deas: it is not possible to open NX designs in I-Deas. Due to this, designing parts and subassemblies in NX would require importing them to I-Deas through neutral file formats, resulting in the loss of traceability, version control and product intelligence.

7.4 Further Development Options

These are further development options which could be pursued but are outside the scope of this thesis. /9/

1. Regarding the long opening time of the full assembly:

    - It may be possible to use the “Named references” function of Teamcenter to allow working with local data. Due to the size of the assembly this may not work.

    - Simplified models could be made and utilized.

    - The customers IM department has suggested a Teamcenter “cache server” to be built at Citec, to allow faster opening time from a server physically closer.

    - The assemblies could be streamlined even more than in the case project of this thesis, which already is very streamlined. This would not have a major impact on the opening time or hardware requirements since the end result will nevertheless be very large.

2. Regarding the high hardware requirements:

    - A similar solution as “Extracted Curves” could possibly be found, to allow removing material from the assembly but still show the related geometry on the drawing. Some stable parts could possibly be made to symbols.

    - The software architecture of I-Deas and NX could be compared. Why does I-Deas handle an assembly of this size flexibly, but NX does not? Perhaps there are software level solutions available.

- Another customer department has also tested using NX for similar work. Their solution could be benchmarked.

3. Parametric design

    - As parametric design is a focus area of the customer while applying NX into design, it would be beneficial for Citec to experiment with implementing parametric design to this product group as well, in co-operation with the customer.

7.5 Recommendation

As stated above, the switch to NX is inevitable. Citec should now decide, in co-operation with the customer, when to do this, and how to address the above listed issues. Information Management department should be included in the decision-making process, due to the hardware implications and the remote desktop solution becoming outdated.

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