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Model Building Techniques for Analysis

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Abstract

The practice of mechanical engineering for product development has evolved into a complex activity that requires a team of specialists for success. Sandia National Laboratories (SNL) has product engineers, mechanical designers, design engineers, manufacturing engineers, mechanical analysts and experimentalists, qualification engineers, and others that contribute through product realization teams to develop new mechanical hardware. The goal of SNL's Design Group is to change product development by enabling design teams to collaborate within a virtual model-based environment whereby analysis is used to guide design decisions. Computer-aided design (CAD) models using PTC's Pro/ENGINEER software tools are heavily relied upon in the product definition stage of parts and assemblies at SNL. The three-dimensional CAD solid model acts as the design solid model that is filled with all of the detailed design definition needed to manufacture the parts. Analysis is an important part of the product development process. The CAD design solid model (DSM) is the foundation for the creation of the analysis solid model (ASM). Creating an ASM from the DSM currently is a time-consuming effort; the turnaround time for results of a design needs to be decreased to have an impact on the overall product development. This effort can be decreased immensely through simple Pro/ENGINEER modeling techniques that summarize to the method features are created in a part model. This document contains recommended modeling techniques that increase the efficiency of the creation of the ASM from the DSM.

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ACRONYMS

ASM	analysis solid model
CAD	computer-aided design
DSM	design solid model
NWC	Nuclear Weapons Complex
SNL	Sandia National Laboratories
WR	war reserve

INTRODUCTION

The practice of mechanical engineering for product development has evolved into a complex activity that requires a team of specialists for success. Sandia National Laboratories (SNL) has product engineers, mechanical designers, design engineers, manufacturing engineers, mechanical analysts and experimentalists, qualification engineers, and others that contribute through product realization teams to develop new mechanical hardware. In the past, the efficiency of these teams was limited, in part, because each individual utilized tools and product data unique to their specialty. The sharing of electronic files was irrelevant because of incompatible formats. Recently, commercial software companies began collaborating to provide integrated suites of engineering applications [2], notably in the areas of design, analysis, and testing, to enable new opportunities for product development teams.

SNL's Design Group is developing model-based engineering services. SNL's goal is to change product development by enabling design teams to collaborate within a virtual model-based environment whereby analysis is used to guide design decisions. SNL believes that the primary sets of data – the foundation for model-based engineering – are three-dimensional solid geometry models. Commonly called the "design definition," these models generally contain the essential product data needed for analytic modeling.

Computer-aided design (CAD) models using PTC's Pro/ENGINEER software tools are heavily relied upon in the product definition stage of parts and assemblies at SNL. CAD models are utilized from the beginning stages of design, in the manufacturing of the product, and also in the analysis of the parts and assemblies [2]. This is known as model-based engineering. The three-dimensional CAD solid model acts as the design solid model that is filled with all of the detailed design definition needed to manufacture the parts. The CAD models are also used in several other areas, such as the machining process and the analysis and simulation process. The CAD design solid model (DSM) is the foundation for the creation of the analysis solid model (ASM). The ASM differs from the DSM in the amount of detailed information that is included. Typically, small features such as rounds, chamfers, and small holes and similar features are not included in the ASM. The creation of the ASM usually requires an analyst to modify the CAD design model to fit certain requirements for the corresponding analysis, and it can be a long, time-consuming process if the DSM is not created using certain modeling techniques. This document contains recommended modeling techniques that increase the efficiency of the creation of the ASM from the DSM. These techniques are defined for the creation of a Pro/ENGINEER DSM to be used for analysis for weapon (war reserve, WR) and non-weapon systems. Within SNL, design models are utilized in the analysis process of the WR and non-WR components and systems. It is the goal of this document, created by the Modeling and Simulation Department at SNL, to create an efficient route between the design and the analysis processes.

An ASM typically does not require some of the detailed geometry created during the design process. Several factors currently exist in a DSM that negatively affects the ASM. Some examples are geometry anomalies, partial product definition, and detail in the model unnecessary for analysis. In order to create an ASM, the model must be manipulated to create a simplified model that is ready for analysis modeling. The simplification process involves an analyst

removing small details that are not needed for the simulation such as holes, rounds, chamfers, and other small details that negatively affect the finite element analysis. If too much detail exists, the analysis process is slower and increases the amount of time to generate a successful simulation. Another difficulty an analyst may encounter is incomplete parameter fields that are required to create a fully defined DSM. The material, density, and mass are some of the parameters that are essential for full traceability between the design definition and the simulation.

The process of creating a design and taking it through analysis can become more efficient if changes are incorporated at the design stage of a Pro/ENGINEER design model. The resulting design model can be meshed and ready for analysis quickly, resulting in saved time, money, and effort. Another benefit of utilizing the DSM for the ASM is the traceability of the parts and assemblies. If an assembly needs to be updated due to new parts or different parts being added, the bulk of the ASM is contained in the design definition Pro/ENGINEER file set, so the work invested in the ASM does not need to be repeated. This also contributes to traceability of the ASM to the DSM. The ASM can be traced to the design CAD model, the design engineer, the list of materials, and to other assemblies, thereby creating a complete set of information.

BACKGROUND

Model-Based Engineering

Engineering at SNL is becoming more dependent on the models from the beginning stages through the end product. The CAD models serve several purposes from visualization, design layout, geometry checks, drawing creation, finite element analysis, and rough machining. The CAD models create a foundation for future functions to create a fully defined, reliable end product.

Model-Based Analysis

The realization of model-based analysis fully relies upon the design definition CAD models. In the Modeling and Simulation Department at SNL, model-based analysis is accomplished through a tool in Pro/ENGINEER that allows an analyst to create an ASM from the DSM. This process uses the simplified representation (rep) tool in Pro/ENGINEER to simplify parts and assemblies for analysis. Using the design definition CAD models ensures the analyst is using the latest design files and also ties the analysis data to the original design data. The analysis is based upon the design CAD models and can be referenced and updated at any time during the analysis process.

With the simplified rep tool, features can easily be excluded, included, or substituted with the click of the mouse. In order for this simplification process to work, the items to be excluded from the part or assembly must have individual features. For example, if a hole is created as a separate feature using a cut or the Hole Tool, the simplified rep method works very easily through selection of that particular cut/hole feature in the model tree. If the hole was created in a sketch, along with a protrusion, the hole is no longer a single feature and is embedded in a protrusion, thereby complicating the process of excluding the hole without excluding the entire protrusion. In this situation, the only means of getting rid of the hole is to create an additional protrusion and fill the hole. This creation of geometry beyond the original design definition alters the model and the traceability. Figure 1 displays a part in the simplification process. The simplified rep tool is capable of excluding the unwanted features such as rounds, chamfers, fillets, holes, etc.

The method in use before the simplified rep process of creating of an ASM required the use of the suppression of features with the creation of new geometry to fill holes or cut away unwanted geometry features. This method was time-consuming and caused models to fail due to the parent/child relationships that exist in a Pro/ENGINEER model. The new method using the simplified rep tool is faster, more efficient, creates traceability, and has become an important method for creating the ASM. One comparison of the two processes showed that the simplified rep method was four times faster than the old method of suppressing features and creating new geometry to fix the analysis solid model.

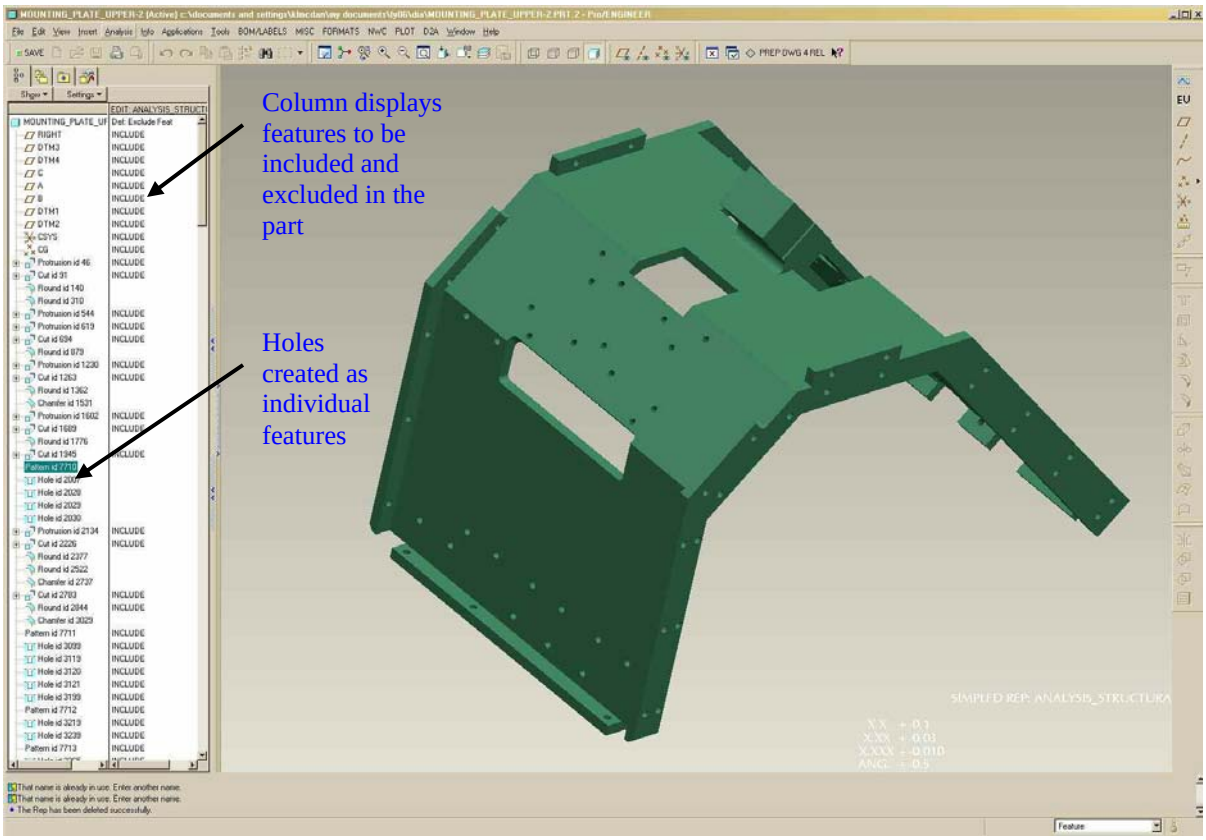

Figure 1. A part in the simplified representation tool in Pro/ENGINEER.

SCOPE

The following techniques can be applied to all new Pro/ENGINEER solid geometry files created for mechanical engineering design and for finite element analysis. The techniques are designed to be incorporated in the beginning stages of modeling along with the recommended start design template. Any mechanical engineering designer creating a new solid model should apply the following techniques to their daily tasks. It is possible that some of the techniques cannot be fully applied. It is suggested that the designer use his/her discretion to apply as many of the techniques as possible, especially if the designer is creating a new part/assembly.

Use of This Document

Mechanical designers should review and refer to this document before generating a new Pro/ENGINEER DSM. In addition to this document, a "Quick Reference Guide" for fast and easy access to the information is located in Appendix A of this document. The goal is to provide useful and important techniques for properly creating and/or updating Pro/ENGINEER DSM for analysis use.

The goal of this document is to have an analyst involved in the initial stages of the design process as opposed to the end of the process when the design definition is complete. The analyst's involvement will help create an optimum design by performing analyses in the beginning, thereby eliminating as much redesign of a part/assembly due to unacceptable structural and/or thermal behavior in normal and abnormal environments.

DISCUSSION

The model building techniques must be addressed by the originator of the data during the development of the design solid models. These areas are covered in more detail in the following sections in order of importance. Each of the outlined techniques is a desired analysis requirement for the designer to apply to the qualifying Pro/ENGINEER files.

Parameters

At SNL, the use of several parameters within a Pro/ENGINEER model is a requirement for the design definition set to be complete. The parameters to be filled out include information that creates traceability and fully defines the parts and assemblies. Several parameters in the model are crucial for the ASM to be complete, such as the materials of the parts. If the parameters are filled in correctly at the beginning of the design stage, this saves time as the information is available and correct once the analyst begins the transformation of the DSM to the ASM. To create a complete traceability package of the CAD models, the most important parameter fields are listed below:

- Official part number, for traceability.
- Bill of materials description name of the assembly or part, traceability of the models.
- Designer's name, and a contact person, for any questions.
- Units of the model, important for analysis.
- Material information of the parts.

The material of the modeled parts should be assigned at the beginning stages of the design. The material properties will then be assigned to the part files and will carry through to the assemblies. Often times, analysis is completed on assemblies that contain more than one part. If the analyst has the material information in the DSM, this decreases the time spent on researching the part for the correct material information; this is particularly true if an assembly has 50 parts or more, which is common for analysis.

Cosmetic Features

Cosmetic features, threads, and notes should be placed on a layer that can be suppressed. Cosmetic features are sketched features that are "drawn" on the surface of a part. Some examples of cosmetic features that may exist in a model are company logos, stamped serial and part numbers, or a feature that is sketched to represent the diameter of a thread. During some geometric translations, cosmetic features are carried as separate entities that are typically not used in analysis. Cosmetic features, sketches, and notes should not have any children defined in the Pro/ENGINEER model. All cosmetic thread features should also be placed in one layer. These cosmetic features should be placed on a layer that can be suppressed to help with the simplification process and the translation process.

Holes, Cuts, Rounds/Fillets, and Chamfers

Holes, cuts, rounds, fillets, and chamfers should be created as separate features, not included as part of a sketch. When creating a hole or cut, create the hole as a single feature by using the Pro/ENGINEER Hole Tool or by creating a separate cut as a feature. For a round or a chamfer, use the corresponding Pro/ENGINEER tool, so that these are defined as individual features and not a part of a sketch. When creating a fillet, use the Pro/ENGINEER fillet tool, not the extrude tool. These entities, as separate features, greatly help the simplification process for analysis. By having features called out by their respective geometric type such as hole, round, or chamfer, an automatic process can be set up to simplify or exclude those geometric entities. If the features are included in a sketch, an additional protrusion has to be created to get rid of the feature as opposed to the quicker selecting of the separate features with the click of the mouse. By creating the new geometry, the design definition is altered from the original design state. Figure 2 shows an unfavorable sketch of an extrusion that has been created to include several features.

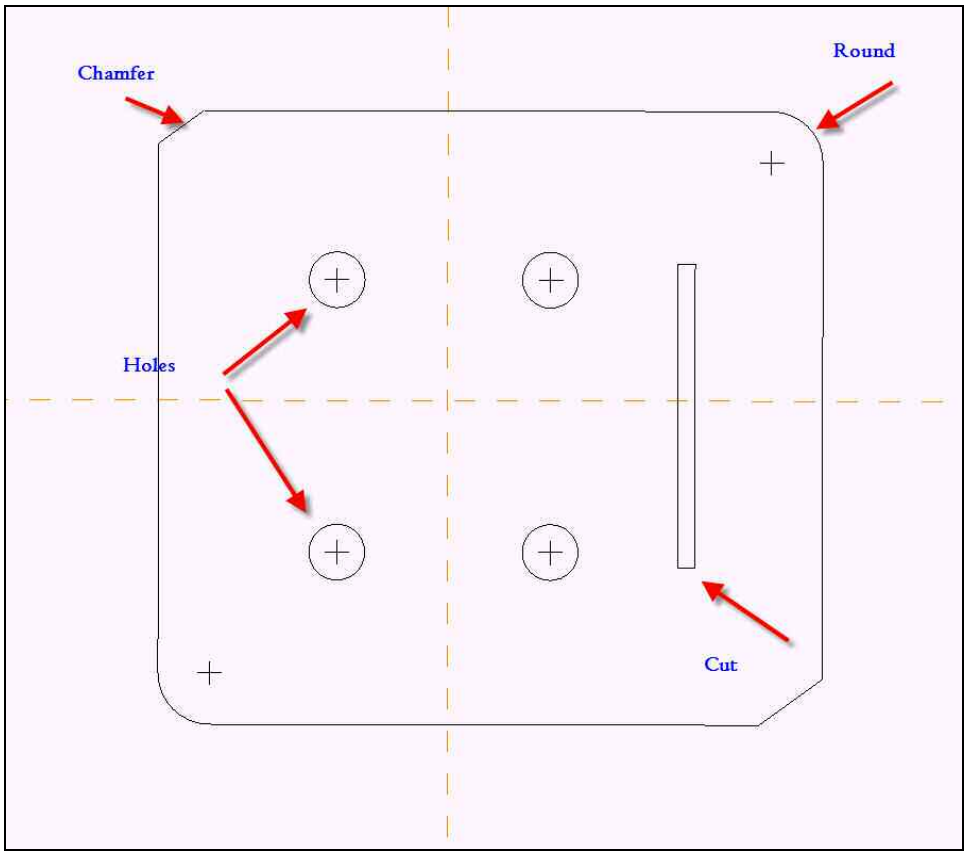

Figure 2. Unfavorable extrusion sketch with several features included.

Additionally, other features that are small relative to the model should be created as separate features also, not within a sketch with other entities (see Figure 3). Communicate with the analyst to determine what is considered a small feature. Ideally, if every geometric entity was an individual feature in a part, then the simplification process would be quicker, easier, and have excellent traceability.

Rounds, fillets, and chamfers should be the last features created in the Pro/ENGINEER part model. Holes should also be created last if they are not a major part of the design definition. This reduces the parent/child relations, improving the simplification process for analysis.

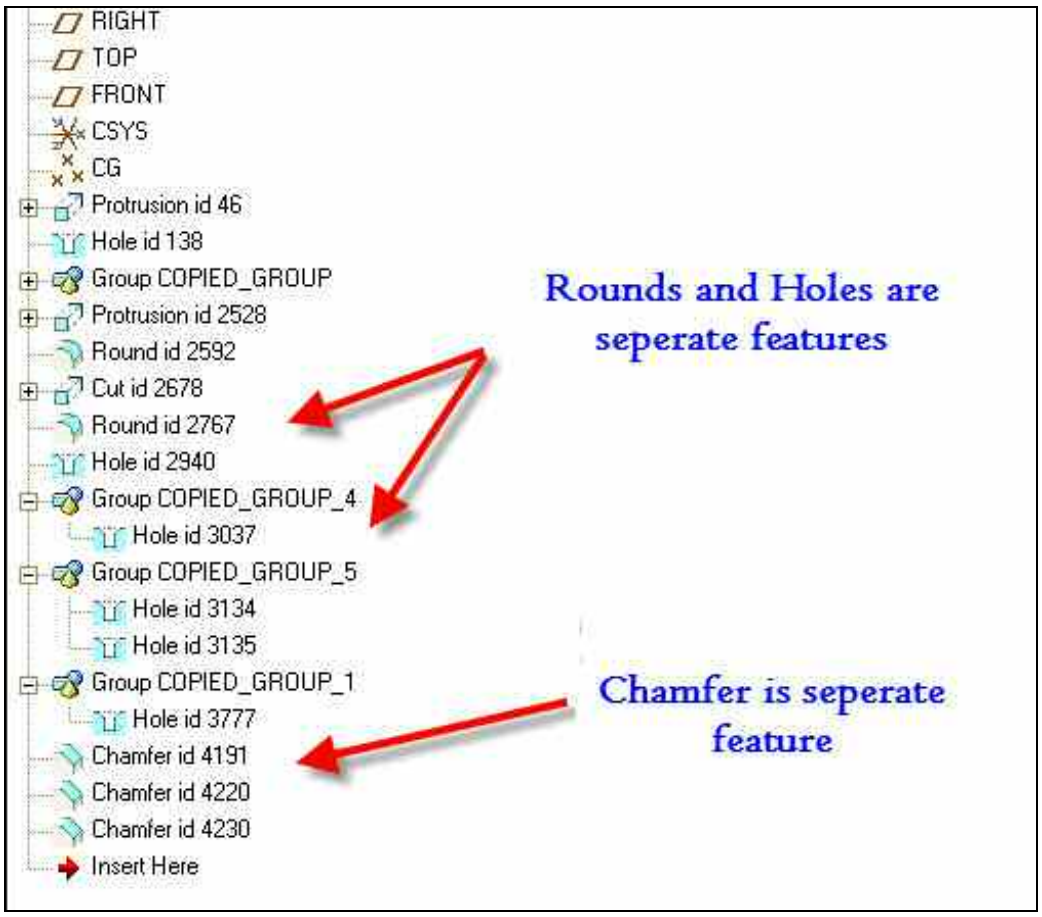

Figure 3. Desired model created using separate features.

Sketched Features

Revolves and extrusions in Pro/ENGINEER are defined using the sketcher and the solid part is developed through an extrusion or revolution of the sketched diagram. Revolves and extrusions should be modeled as simply, including as few features in the sketch as possible. Avoid creating multiple features in one sketch as this will help with the simplification process. For complicated revolves, break the part into several revolved sections, with one section contained in a separate sketch. For example, when creating a pulley that has many different levels and grooves, the part can be created with one sketch; but, for easier feature exclusions, each notch can be defined as a separate entity. If the analysis needs to remove one groove or level, then the entire part will not be suppressed: each groove can be excluded independently instead. The objective is to build a complicated revolve or extrusion using several features as opposed to one sketch that defines the entire part. Figure 4 shows the different grooves and multi-level surfaces that can be modeled separately in a part. If this part was created with one revolve that included the grooves and different levels in the revolve sketch, the simplification process would not work without excluding the entire part in the simplified rep.

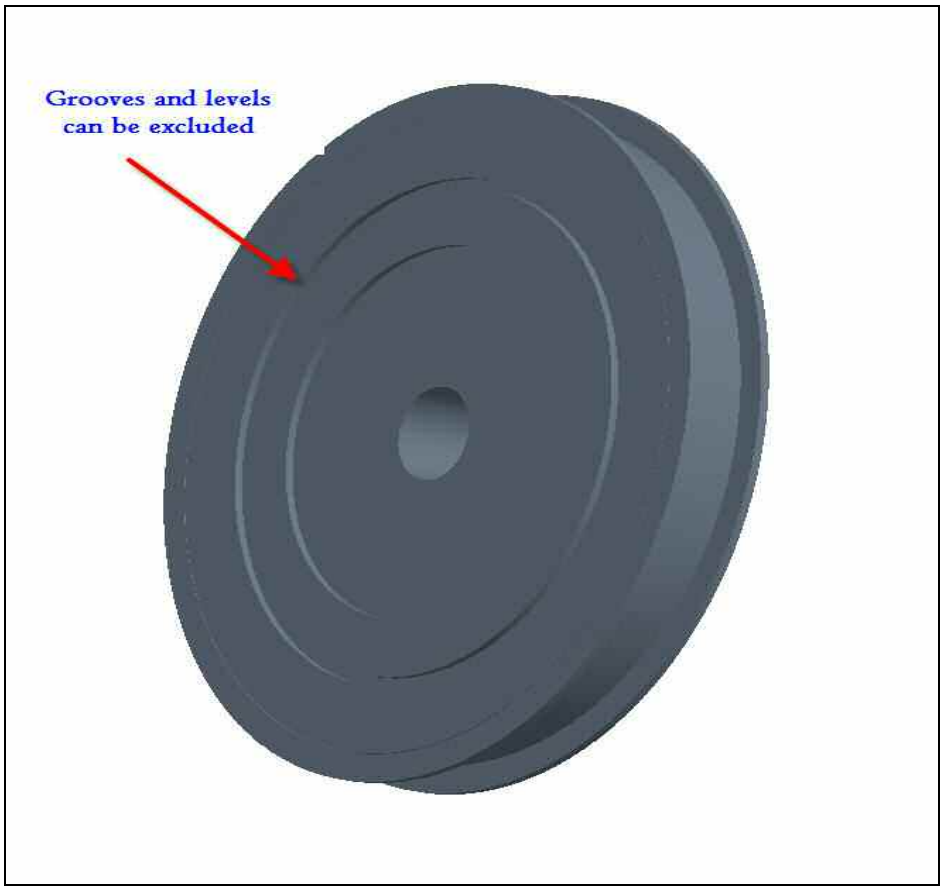

Figure 4. Pulley with grooves and different level surfaces

Part Geometry and Assembly Interfaces

Special care is required in modeling of the parts to help with the analysis model conversion. Assembly interfaces are important in an ASM. While modeling, special attention is needed to eliminate unwanted interferences and gaps. Interferences and gaps between parts in an assembly are problematic for creating the ASM. While modeling, it is recommended that the designer check global interferences on assemblies before sharing models for analysis. If interferences exist, examine and correct the intersecting parts. Some exceptions exist for interferences in the CAD model. Press fit parts such as fasteners, helicoils, and pins are acceptable interferences. Other exceptions to the interference recommendation are preloaded parts, o-rings, compression pads, and malleable materials. Global interferences are acceptable with these listed parts. If gaps exist between parts that are not a part of the design definition, investigate and correct the existing gap. When possible, make a Pro/ENGINEER Instance of the assembly. Typically, an analysis model requires the deformed (as-assembled) state of all parts. Another recommended method is to model to the mean of the part dimensions. Mean modeling is becoming the norm for modeling of parts because of the benefits during the manufacturing process.

Accuracy

At SNL, the default accuracy of the Pro/ENGINEER models is defined to be 1e-4 (0.0001). With an ASM, the models are translated into different software for pre-processing. Sometimes the translation can be difficult due to different accuracy settings between the two software packages. If the designer is currently working with analysis and the Pro/ENGINEER files are creating problems during the translation, attempt to regenerate the model(s) with an absolute accuracy less than the current default setting. Occasionally, if the model(s) fail regeneration with an accuracy level of 1e-6 (0.000001), the cause of the regeneration problem results in inaccurate geometry during the translation.

External References/Copied Geometry

Geometry created from external references should be removed when possible. Any external reference relationships in a model must be broken on features. External references are created from the use of molds, cast parts, merge functionality, external copy geometry, and features dependent on other features or models outside the current model. These dependencies create relationships between features, hindering the simplification process for analysis. When possible, avoid using external references when creating a model. In addition to external references, copied geometry should be set to "independent" so as not to complicate the simplification process.

Assembly Features

Avoid assembly features if possible. Features should not be created at the assembly level. If an assembly cut, hole, or other feature is required, activate the part or parts that require altering at the assembly level and create the desired features. Make sure the referenced features created in this manner are within the scope of the part and not entities outside of that part such as assembly edges, faces, or points. This technique helps maintain the traceability and integrity of the part files. Assembly features are difficult to exclude in the simplified rep process and can be missed when viewing affected parts at the part level instead of the assembly level.

Foams and Encapsulates

All existing parts in an actual assembly should be modeled. At SNL, one common practice is to not model fillers, such as foams, that are present in an assembly. Foams and encapsulates should be modeled where used. Foams and encapsulates are not required to be one part for analysis. Multi-body parts or multiple parts to represent a foam or encapsulate used to fill a cavity are acceptable for analysis. Refer to the SNL Modeling and Simulation Department for further assistance when creating foams and encapsulates using Pro/ENGINEER.

Shrink-Wrap

The Pro/ENGINEER operation known as shrink-wrap is used to help reduce the memory used in a part or assembly by including only the exterior shape of a model. Avoid the use of shrink-wrap parts for models; make original part files available when possible instead. When a shrink-wrap is included in an assembly, the simplified rep process does not work since the entire part or

assembly is listed as one feature. Also, avoid making one Pro/ENGINEER part resemble many parts. If shrink-wrap is needed for a design, create a simplified rep called Shrinkwrp01, where the number, 01, will iterate based on the number of shrink-wrap representations needed. The master representation will contain all original design parts that can be used for an analysis representation of the model. If a shrink-wrapped file must be supplied to the analyst, include as much product definition as possible in the Pro/ENGINEER file. If the creation of a shrink-wrap cannot be avoided, fill out the parameter fields to create traceability for the Pro/ENGINEER file.

Vendor Parts

Not all parts are designed, modeled, and manufactured at SNL. For off-the-shelf parts, the vendors may supply the drawings or the CAD models of their parts or assemblies. The vendor CAD models require special care to include them in the design definition of SNL parts and assemblies. All off-the-shelf parts should follow the proper naming convention for Commercial Supplier / Commercial Product CAD files received from a vendor. All Pro/ENGINEER files obtained from vendors should have a complete product definition similar to the SNL created design models.

Units

All models should use consistent units within an assembly when possible. When creating an assembly of several Pro/ENGINEER files, the designer should ensure that the units are consistent throughout every part. If the units are not consistent, this will cause problems with the translation of the geometry into the pre-processors.

CONCLUSIONS

Methods to develop Analysis Solid Models (ASMs) from Design Solid Models (DSMs) were described in this report. These methods are under constant evaluation and improvement. However, they have been tested and validated on many full-system and component finite element analysis modeling projects.

Retaining the connectivity between a new ASM and the design model is a goal with several benefits. One benefit is the traceability of the ASM to the Pro/ENGINEER design definition. Another benefit, discussed in detail throughout this report, is a quicker and more efficient transformation between the DSM and the ASM.

The techniques recommended herein help to promote traceability and efficient data transformation between the design and analysis communities. The techniques are based on the Simplified Representation Tool in Pro/ENGINEER. This "Simplification Rep" process has been shown to be faster and more efficient than older processes, utilizing suppression of features and creation of specialized geometries to "fix" the CAD models. The older processes negatively impacted the design definition files by actually altering design data. The new method, using simplified reps, preserves the design definition as much as possible.

Model-based engineering is a SNL goal. The design and analysis teams are working together to create high-quality, fully defined design definition three-dimensional CAD models that can be utilized in other areas of engineering. This document was created to promote the creation of ASMs and to communicate these best practices as recommended techniques. The mechanical designer must use their experience and judgment to apply as many of the techniques as possible on every design project.

RESOURCES

[1] MSC Software Corporation, www.mscsoftware.com/

[2] Rick Harris and Jack R. Martinez, Model-Based Engineering, SAND2007-2770, Sandia National Laboratories, Albuquerque, NM, May 2007.

[3] Jack R. Martinez, Brandon Moore, and Stephanie L. Pollice, SNL Mechanical Computer Aided Design (MCAD) Guide – 2007, SAND2007-8071, Sandia National Laboratories, Albuquerque, NM, December 2007.

APPENDIX A. QUICK REFERENCE GUIDE

This guide can be cut and detached along the perforation marks. There are three copies for your use.

Quick Reference Guide – Rev. II
Pro/ENGINEER Model Building Techniques For Analysis

Always create new parts with a start part file. Follow all techniques in this guide as closely as possible.

Section 1: Parameters

Complete all NWC parameters listed below.

- PART_NO, the official NWC part number (i.e., 310310-00).
- BOM_DESCRIPTION_ASM, the descriptive part name (i.e., BRACKET).
- MATERIAL, the material description (i.e., ALUM, 6061-T6).
- PART_SPECIFICATION_NO, the material specification (i.e., 7420301-02).
- UNITS, the units used within the geometry model (i.e., INCH).
- ISSUE, the issue of the released geometry model (i.e., A, B, C, etc.).
- PART_CLASSIFICATION**,** the classification of the part, subassembly, or component.
- CHECKED and APPROVED (i.e., Model check and designer).

Section 2: Cosmetic Features

Cosmetic features and sketches should not have children.

- Place on a layer that can be suppressed.
  - Threads: _2_thread layer (start part default)
  - Sketches: _2_cosm_sketch layer (start part default)

Section 3: Holes, Cuts, Rounds/Fillets, and Chamfers

Holes, cuts, rounds/fillets, and chamfers should not have children.

- Create as separate features. Do not include in a single sketch, especially if they are small relative to the model.
- Use the appropriate feature creation tool.
- Reference these features to large entities that will most likely not be removed such as the default datum planes.

Section 4: Sketched Features

- Avoid small details in revolved and sketched features.
- Place single features in one sketch.

Section 5: Part Geometry and Assembly Interfaces

- Check global interferences on assemblies before sharing models for analysis. If gaps or interferences exist that should not be there, examine and correct.
  Exceptions include:
  - Press fit parts or hardware (i.e., fasteners, inserts, and pins)
  - Preloaded parts (i.e., o-rings, compression pads, malleable materials, etc.)
    - Create the deformed part.

Quick Reference Guide – Rev. II
Pro/ENGINEER Model Building Techniques For Analysis

- Geometry Construction – Use mean dimensioning.
- Gaps – Investigate and correct unintentional gaps (i.e., assembling parts using an offset coordinate system).

Section 6: Accuracy

- Attempt to regenerate the model at an absolute accuracy of 1e-6.
- All Pro/ENGINEER files should not have a default absolute accuracy larger than 1e-4. (This is a Wildfire 2.0 default.)

Section 7: External References/Copied Geometry

- Do not use external references. External reference relationships should be broken by redefining the part.
- Set copied geometry to "independent."

Section 8: Assembly Features

Cuts should not be created at the assembly level.

- Referenced features should be kept within the scope of the part. Do not reference entities outside that part such as assembly edges, faces, or points.

Section 9: Foams and Encapsulates

Model foams and encapsulates where used.

- Multi-body parts or multiple parts to represent a foam or encapsulate used to fill a cavity are acceptable for analysis.

Section 10: Shrink-Wrap

Supply original parts files to analyst if possible. Do not use shrink-wrap parts for models.

- If shrink-wrap must be used for a design, create a simplified representation called Shrinkwrp01, where the number, 01, will iterate based on the number of shrink-wrap representations needed.
  - Include as much product definition as possible in the Pro/ENGINEER file.
  - Fill out the NWC parameter fields to create traceability for the Pro/ENGINEER file.

Section 11: Vendor Parts

Follow the naming convention within MCAD Design Guide for all off-the-shelf parts. Files obtained from vendors should have a complete product definition as per NWC design model standards.

Section 12: Units

Do not intentionally mix units in an assembly. Use consistent units within an assembly when possible.

Contacts

Modeling & Simulation, Organization 02991
Howard Walther, (505) 844-2224

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Pro/ENGINEER Model Building Techniques For Analysis

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- PART_CLASSIFICATION**,** the classification of the part, subassembly, or component.
- CHECKED and APPROVED (i.e., Model check and designer).

Section 2: Cosmetic Features

Cosmetic features and sketches should not have children.

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  - Threads: _2_thread layer (start part default)
  - Sketches: _2_cosm_sketch layer (start part default)

Section 3: Holes, Cuts, Rounds/Fillets, and Chamfers

Holes, cuts, rounds/fillets, and chamfers should not have children.

- Create as separate features. Do not include in a single sketch, especially if they are small relative to the model.
- Use the appropriate feature creation tool.
- Reference these features to large entities that will most likely not be removed such as the default datum planes.

Section 4: Sketched Features

- Avoid small details in revolved and sketched features.
- Place single features in one sketch.

Section 5: Part Geometry and Assembly Interfaces

- Check global interferences on assemblies before sharing models for analysis. If gaps or interferences exist that should not be there, examine and correct.
  Exceptions include:
  - Press fit parts or hardware (i.e., fasteners, inserts, and pins)
  - Preloaded parts (i.e., o-rings, compression pads, malleable materials, etc.)
    - Create the deformed part.

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Section 7: External References/Copied Geometry

- Do not use external references. External reference relationships should be broken by redefining the part.
- Set copied geometry to "independent."

Section 8: Assembly Features

Cuts should not be created at the assembly level.

- Referenced features should be kept within the scope of the part. Do not reference entities outside that part such as assembly edges, faces, or points.

Section 9: Foams and Encapsulates

Model foams and encapsulates where used.

- Multi-body parts or multiple parts to represent a foam or encapsulate used to fill a cavity are acceptable for analysis.

Section 10: Shrink-Wrap

Supply original parts files to analyst if possible. Do not use shrink-wrap parts for models.

- If shrink-wrap must be used for a design, create a simplified representation called Shrinkwrp01, where the number, 01, will iterate based on the number of shrink-wrap representations needed.
  - Include as much product definition as possible in the Pro/ENGINEER file.
  - Fill out the NWC parameter fields to create traceability for the Pro/ENGINEER file.

Section 11: Vendor Parts

Follow the naming convention within the MCAD Design Guide for all off-the-shelf parts. Files obtained from vendors should have a complete product definition as per NWC design model standards.

Section 12: Units

Do not intentionally mix units in an assembly. Use consistent units within an assembly when possible.

Contacts

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Quick Reference Guide – Rev. II
Pro/ENGINEER Model Building Techniques For Analysis

Always create new parts with a start part file. Follow all techniques in this guide as closely as possible.

Section 1: Parameters

Complete all NWC parameters listed below.

- PART_NO, the official NWC part number (i.e., 310310-00).
- BOM_DESCRIPTION_ASM, the descriptive part name (i.e., BRACKET).
- MATERIAL, the material description (i.e., ALUM, 6061-T6).
- PART_SPECIFICATION_NO, the material specification (i.e., 7420301-02).
- UNITS, the units used within the geometry model (i.e., INCH).
- ISSUE, the issue of the released geometry model (i.e., A, B, C, etc.).
- PART_CLASSIFICATION**,** the classification of the part, subassembly, or component.
- CHECKED and APPROVED (i.e., Model check and designer).

Section 2: Cosmetic Features

Cosmetic features and sketches should not have children.

- Place on a layer that can be suppressed.
  - Threads: _2_thread layer (start part default)
  - Sketches: _2_cosm_sketch layer (start part default)

Section 3: Holes, Cuts, Rounds/Fillets, and Chamfers

Holes, cuts, rounds/fillets, and chamfers should not have children.

- Create as separate features. Do not include in a single sketch, especially if they are small relative to the model.
- Use the appropriate feature creation tool.
- Reference these features to large entities that will most likely not be removed such as the default datum planes.

Section 4: Sketched Features

- Avoid small details in revolved and sketched features.
- Place single features in one sketch.

Section 5: Part Geometry and Assembly Interfaces

- Check global interferences on assemblies before sharing models for analysis. If gaps or interferences exist that should not be there, examine and correct.
  Exceptions include:
  - Press fit parts or hardware (i.e., fasteners, inserts, and pins)
  - Preloaded parts (i.e., o-rings, compression pads, malleable materials, etc.)
    - Create the deformed part.

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- Geometry Construction – Use mean dimensioning.
- Gaps – Investigate and correct unintentional gaps (i.e., assembling parts using an offset coordinate system).

Section 6: Accuracy

- Attempt to regenerate the model at an absolute accuracy of 1e-6.
- All Pro/ENGINEER files should not have a default absolute accuracy larger than 1e-4. (This is a Wildfire 2.0 default.)

Section 7: External References/Copied Geometry

- Do not use external references. External reference relationships should be broken by redefining the part.
- Set copied geometry to "independent."

Section 8: Assembly Features

Cuts should not be created at the assembly level.

- Referenced features should be kept within the scope of the part. Do not reference entities outside that part such as assembly edges, faces, or points.

Section 9: Foams and Encapsulates

Model foams and encapsulates where used.

- Multi-body parts or multiple parts to represent a foam or encapsulate used to fill a cavity are acceptable for analysis.

Section 10: Shrink-Wrap

Supply original parts files to analyst if possible. Do not use shrink-wrap parts for models.

- If shrink-wrap must be used for a design, create a simplified representation called Shrinkwrp01, where the number, 01, will iterate based on the number of shrink-wrap representations needed.
  - Include as much product definition as possible in the Pro/ENGINEER file.
  - Fill out the NWC parameter fields to create traceability for the Pro/ENGINEER file.

Section 11: Vendor Parts

Follow the naming convention within MCAD Design Guide for all off-the-shelf parts. Files obtained from vendors should have a complete product definition.

Section 12: Units

Do not intentionally mix units in an assembly. Use consistent units within an assembly when possible.

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