AUTODESK UNIVERSITY

UT20934

3D Modeling Practices and Standards for Substations

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Learning Objectives

- Learn how to produce 3D models that are accurate, consistent, & user-friendly
- Discover best practices for part and assembly creation
- Learn how to use Vault to capitalize on 3D model data
- Learn how to use Inventor to transform the substation design process

Description

This class will cover industry best practices and workflows for 3D substation design projects utilizing Inventor Professional software and the Substation Design Solutions toolkit. In addition, we will cover integration of Vault Professional software to maximize the efficiency of the 3D modeling processes. We will detail processes and procedures to ensure consistency, flexibility, and overall usability of 3D models, and we'll learn how to change the way substation designs are currently done. We will present Vault categories, lifecycles, and states as a means to demonstrate utilization of 3D model data and how it can be integrated into existing systems to facilitate custom reporting and data analytics. This session features Inventor Professional and Vault Professional.

Your AU Expert(s)

Mr. Chasser works for Burns & McDonnell Engineering as the Substation Design Section Manager for the World Headquarters office in Kansas City, MO. He has 12 years of substation design experience ranging from 12kV-765kV. Mr. Chasser has 2+ years of experience utilizing Autodesk Inventor & Vault, along with the Substation Design Suite toolkit.

Mr. Rugg works for Burns & McDonnell Engineering as an Electrical Detailer in the Transmission & Distribution Department in Kansas City, MO. He has 12+ years of experience using Autodesk Inventor & Vault, across multiple industries. Mr. Rugg has 2+ years of experience utilizing the Substation Design Suite toolkit. He is an Autodesk Inventor Certified Professional.

Learn how to produce 3D models that are accurate, consistent, & user-friendly

Creating models that are accurate, consistent, & user-friendly is the goal of every modeler, sometimes even to a fault. Learning to strike the appropriate balance of all these items is critical to the success (or lack thereof) of any given project. When a company decides to transition to an intelligent 3D design environment it is helpful to look at the end usage of the models and then determine how they will be developed & used.

Begin at the End…

Examining the end usage case of the models is a great way to answer many questions that often derail a 3D implementation. Determine the following:

1.) What will the 3D models be used for?

a. Is the goal to simply streamline the design process, or do you want to take it further? Perhaps your company is consistently building substations in strictly confined areas, and clearance checking is vital. Looking at the end usage goal of the models will help define processes, answer questions, and drive workflows later on down the road. Keep these end usage goals in mind!

2.) What data do we want from the 3D models?

a. The opportunity for data entry & capture is nearly endless. Understand what data you want to get FROM the models in the end, so you know what you'll need to put IN, at the beginning.

3.) Who will be using the 3D models?

a. Understanding the end-user of the models is a key component to understanding how the models should initially be developed. If the end-users will generally be those with limited experience, ensure your models are developed in a manner that allows them to be used/manipulated easily. If your end-users will be experienced your models should still be developed to be easily used/manipulated, however, they can contain more advanced features such as iLogic.

Make Decisions

How much detail is too much? How much is not enough? These are oftentimes the two major questions that arise when deciding how to develop 3D parts/assemblies. Ultimately, this comes down to a "Visual Fidelity vs. Visual Identity" discussion. Visual Fidelity refers to how a particular model resembles its real-world counterpart. Visual Identity refers to if the model can be readily identified as its real-world counterpart. Generally, a low visual fidelity leads to a low visual identity. If a model is developed with a very high visual fidelity (and thus a high visual identity) it generally takes longer to develop & check (QA/QC) & comes with a large number of features & faces that Inventor must calculate, potentially hindering computer performance in large complex assemblies.

After determining how much detail is appropriate for your organization, comes the task of figuring out what models actually need to be developed. This can come in the form of a large ERP database type "data-dump", a carefully constructed list of currently used parts/pieces, or anything in between. Circle back to the beginning and determine the end usage case for each

model. Is it a part that is used very sparingly? If so, can it be developed at a later date, on an "as-needed" basis? How comprehensive do you want your parts library to be? Explore Content Center for some items that may exist in your list, but already exist in Inventor. Determine if you want your parts to be tied to a custom content center library.

The next step after determining how much detail the models should contain and what models are to be developed is to start modeling, correct? Maybe… Consider what your execution plan is for the model development. One way to approach this could be to develop your entire "list" or library of parts up-front, prior to internal/external roll-out of the software. This approach can use internal and/or external resources to develop a comprehensive library of parts that can be used in project execution. An alternate approach is to develop the guidelines by which your parts/assemblies are to be developed, and have them developed on an "as-needed" basis throughout the normal project lifecycle. Over time, this process develops the same comprehensive library. Consider the pros/cons for each method & tailor it to your organization's needs.

One of the most important aspects of 3D substation design utilizing Autodesk Inventor is the guidelines by which your models are developed. Develop guidelines for not only part & assembly modeling, but also consider developing guidelines for the following:

1.) Foundations
2.) Structures
3.) Conduit
4.) Trench
5.) Drawing Output/Creation
6.) BOM Output

Determine how your models will be tracked/organized/controlled. Will this be done through Autodesk Vault, or by other means? It is vital to know what system will be used to store your models and how you can manage the data from your data repository.

Develop standard file naming structures for parts and assemblies. It is best to keep the naming structure simple, but meaningful. The user should be able to read the filename and have a general idea of what the part and/or assembly is, without having to open the file to see what it is or decoding using a separate document.

Discover best practices for part and assembly creation

Implementing best practices is a good starting point on the road to intelligent 3D substation design. Using the following and tailoring them to your organization's specific needs will help leverage industry lessons learned and effectively implement Autodesk Inventor.

Develop Template Files

The first step to implementing Autodesk Inventor is to develop a complete Inventor project file (.ipj). The project file is key to the success of any 3D implementation. The project file provides Inventor the location of template files, base settings, Vault information, etc…

Following the creation of the project file, template files should be developed. These template files should contain all custom iProperties that are needed, as well as the setting of proper units, UCS orientation, styles, and any other information desired. Ensure the creation of the template

files is well thought-out. Changes to template files at a later date creates inconsistencies, which can cause issues down the road.

Z-Axis Should Always Be Up

When developing template files, or any new files for that matter, the Z-Axis should always go up. By standardizing this, it leads to consistency when placing parts in assemblies, and ensures that logical constraints to base work-planes can be used.

Sketches Should Be Fully Constrained

As a general rule, all sketches should be fully constrained. Unconstrained sketches lead to poor model integrity and issues when creating or modifying features. Use dimensionalparameterized constraints as necessary to constrain sketches. Avoid using the "auto-constrain" feature to make sketches fully constrained. Using this feature often leads to confusing constraints that are difficult to use in making adjustments when necessary.

Fill Out iProperties While Developing Parts

It is helpful to fill out all required iProperties (native or custom) during the part/assembly development phase. While it is not impossible to fill them out later, it is good modeling practice to fill these out while you are working on the model. Doing this leads to consistency & ensures that the model data is available at any time and can be extracted when needed.

Use Base Origin Point/Planes/Axis' For Constraints

Take advantage of the base origin point, planes (XY, XZ, YZ) and axis' (X, Y, Z) for constraints in assemblies. Since these are fixed entities within Inventor, it is best to constrain to these when applicable (especially for the first or "base" part in assemblies. Additionally, using "like" constraints such as XY to XY mate or YZ to YZ flush, provides a way to easily make adjustments when necessary. This allows less experienced Inventor users to identify and fix potential constraint issues with greater ease and less time required. **DO NOT** constrain to features unless absolutely necessary.

Utilize End of Part (EOP) Marker to Control Feature Visibility

When developing parts, always develop the "base" shape or figure first, and add work features last. Doing so allows the user to slide the EOP marker up in the browser tree and control feature visibility (visual fidelity). In addition, the user can move the EOP marker up to control file size for performance, storage and or file transfer.

Minimize Use of Patterns & Mirrors

Many Inventor users have been trained to use patterns and mirrors as a means of efficiency in assembly modeling. While this IS a more efficient modeling practice it does have a few drawbacks when it comes to large complex models such as substations. The first drawback is that using takes up a great deal of computer resources (CPU usage), primarily in the larger assemblies. This is due to the manner in which Inventor processes and calculates mirrors and patterns. By minimizing (or completely eliminating) the use of patterns and/or mirrors in assembly files, the user is making the model less resource intensive and easier to use in the end. There are two good alternatives to using patterns and mirrors. The first involves inserting multiple instances of the part and constraining each of them individually. Recognizing this is a slightly less efficient modeling practice, the plus side of this method is that it "forces" the user to

interact with each component of the assembly as an individual entity, much like what is actually done during construction. This is a form of QA/QC that can help identify potential construction issues during the engineering phase, rather than in the field. The second method takes the first method one step further and involves "grounding" each of the parts after constraining them. Once a component is grounded, Inventor ignores (does not process) the constraints for the part/assembly that is grounded. This gives the user flexibility to un-ground and edit the constraints later (if necessary). Remember to re-ground after adjusting the constraints.

Verify Degrees of Freedom
While developing an assembly, routinely check the Degrees of Freedom. This tool shows both translational and rotational degrees of freedom. Ensure that all degrees of freedom are "solved" leads to greater model integrity and reduces the likelihood of parts being able to be inadvertently moved or rotated during development. Also, routinely checking for degrees of freedom allows the user to identify issues as they arise, instead of creating a compounding problem that is more complex to solve as the assembly size grows.

Use Analyze Interference Tool
Utilizing the "Analyzing Interference" tool during assembly development allows the user to readily identify areas in which parts are overlapping each other or otherwise interfering. This helps identify potential construction issues before they reach the field.

Modular Design Approach
Deploying a "modular" design approach is a key component to developing large assemblies that are user-friendly, yet also strike a balance with reducing computer performance issues. Understanding how Inventor processes and calculates assemblies and its associated constraints, as well as sub-assemblies and their associated constraints further proves this best practice. To develop a modular design approach, the user starts with basic part development. These parts are then put together into component assemblies (Example: disconnect switch with associated structure and foundations). Once the component assemblies are developed, the user can place multiple component assemblies into a sub-assembly (Example; substation bay). The final step is to place these sub-assemblies into a master assembly (Example: complete substation model). The development of the component assemblies and sub-assemblies are the key to the modular design approach. These are often "standard" assemblies that are used in many situations, across many substations or switchyards. The user gains a high level of 3D design efficiency by developing & then deploying these assemblies.

Utilize Level of Detail (LOD)
Levels of Detail can be used to help many different aspects of 3D substation modeling. It is recommended that "standard" levels of detail are set up in all necessary template files and are named consistently. By doing this, the user can later use "linked level of details" to even further maximize the functionality of this design tool. Level of detail can be used to suppress items that are not desired for certain views (grounding plans, foundation plans, conduit plans, etc…).

Keep Browser Tree Organized
With large & complex models it can often be cumbersome to find parts/assemblies within your browser tree. Consider creating folders under your browser tree to help with organization and

finding parts/assemblies when needed. These folders can also be set up as a "standard" within the template files.

Use "Enforce Unique Filenames" Feature within Vault

If your standard part/assembly naming convention allows, consider using the "enforce unique filenames" feature. This prevents files with duplicate filenames being placed into the same vault. This feature is designed to help prevent duplication of models, and thus lost time and efficiency due to modeling the same part multiple times.

Learn how to use Vault to capitalize on 3D model data

Autodesk Vault can be an incredibly powerful tool to manage, manipulate, and report on data from 3D models. It can also be used as a data repository that tracks links, relationships, and historical data.

Vault Setup

One of the key elements to utilizing Vault is to determine what users have what particular set of permissions, similar to any standard network system permissions. In Vault these permissions are controlled by user-groups. Each organization should take a holistic look at all users who may potentially have access to their Vault and divide them into the appropriate user-groups based on level of access desired. Next, permissions are granted on a user-group basis.

Determining a proper folder structure is also a key component to properly using Vault to its full potential. Strive to keep the folder structure as "flat" as possible. If you have developed a large, complex folder structure, it is likely you will run into errors when trying to utilize the "pack-and-go" feature from within Vault, running near or over the character limit on file paths.

Lifecycles are a great way to utilize Vault to drive QA/QC processes. The Vault administrator can set up transitions from work-state to work-state within each lifecycle that mimic internal QA/QC processes and require that certain parameters be met to grant permission for the file(s) to transfer work-states.

Vault categories can be used to organize files & enforce adherence to policies & procedures. These can be helpful when different lifecycles are needed within an organization. If different lifecycles are necessary, set up the appropriate number of categories, and assign lifecycles to the categories as applicable.

Copy Design

Copy design is a powerful tool within Vault. Use copy design to create "like" or similar files from a base set of files. This is similar to doing a "save-as" in many other programs, but with added functionalities such as reuse vs. copy, automatic prefix of suffix, and incrementing tools.

Managing Information

Vault makes managing 3D data and model information simple. Tasks such as mass iProperty editing can be done on large datasets. The advanced search feature allows the user to search on nearly any criteria, define where a particular search should occur, or search on user-defined

criteria. Vault provides file tracking capabilities that allow the user to see all instances of where the particular file is being used and also see the progressive history and versions of all files.

Vault Reporting

Vault allows the user to run base or custom user-defined reports on a wide range of criteria. These reports can be fully customized, or Vault comes pre-loaded with a set of report templates.

Learn how to use Inventor to transform the substation design process

Autodesk Inventor as a stand-alone tool is very powerful. Couple this tool with SBS Substation Design Suite toolkit add-on and the possibilities are endless. The Substation Design Suite toolkit add-on brings substation specific tools to the Inventor platform. These tools are comprised of above-grade, below-grade, and design checking tools aimed at streamlining processes and aiding in substation checks and calculations.

Above Grade Tools

Utilizing the above grade tools menu within the SDS toolkit allows the user to insert terminal, cable, & bus fittings. The tools allow for bare cable and rigid bus routing through these fittings. The fitting library can be customized through the SDS Content Editor to fit any organization's specific needs. The above grade tools also allow for a graphical representation of lightning protection analysis.

Below Grade Tools

The below grade tools menu allows the user to efficiently populate a ground grid, populate a cable trench run, and route conduits. These tools are also customizable through the SDS Content Editor.

Design Check Tools

Clearance checking can be done efficiently and accurately using the design check tools. Users can check phase-to-phase, phase-to-ground, fence safety, & live part-to-roadway clearances. With minimal input, all clearances can be verified and validated. The clearance values are customizable through the SDS Content Editor.

AutoCAD Electrical Integration

Inventor 2016 (and later releases) support an electromechanical link (.emx file) between Inventor and AutoCAD Electrical. This electromechanical link provides cross-discipline checking with protection & control drawings. Schematic diagrams can be checked against the physical representations in Inventor and vice-versa.