An Evaluation of the Pro/ENGINEER Associative Interface for Abaqus/CAE

Jerome Montgomery

Siemens Power Generation Inc., Orlando, FL.

Abstract: *As design cycle times are shortened, engineers continue to find ways to be more productive. Generally, one tries to leverage software tools to get solutions faster. This productivity improvement is possible through continued software advancements, such as the Pro/ENGINEER to Abaqus/CAE Associative Import, via an Elysium Neutral (Assembly) File. Engineers can iterate from CAD to FEA while retaining applied loads and boundary conditions. This paper will discuss use of the associative import features with design changes (such as thickness change, split surfaces, or holes). The pros and cons in using the tool will be tabulated. The reader will see that using the associative import can reduce design iterations from weeks to days.*

Keywords: *Pro/ENGINEER, Pro/E, Pro, Associative Interface, Interface, Abaqus/CAE, Abaqus, CAE, FEA, Finite Element, Finite Element Modeling, CAD, Automatic Associative Import, Manual Associative Import, Direct Translator, NonAssociative.*

1. Introduction

Recurring issues in the finite element analysis (FEA) world has been retrieval of clean geometry, and geometry creation limits in FEA software. Since geometry creation is not a specialty of finite element software, limited emphasis has been placed on the geometry engine. Most emphasis, as expected, has been on the analysis engine (mesh generators and solvers).

Rightfully, computer aided design (CAD) software has specialized in geometry creation and have remained the primary software for this arena. CAD software companies are used by large corporations because of their CAD capabilities in geometry creation, modeling, and their ability to connect to other software.

Pro/ENGINEER (Pro/E) is one such CAD software company that provides geometry based capabilities needed for FEA connectivity by Abaqus analysis software. But like other FEA companies, the concern is that the CAD package and the FEA package are not linked. Once the geometry is created in the CAD software, it is important to import the geometry data into the FEA software without losing geometry features.

This has been accomplished by translators developed by different companies. The translators are either connected or unconnected mode translators. They maintain a connection between the CAD software and the FEA software or they do not. Sometimes the terms plug-in (connected) or reader

(unconnected), are used. The term used in Abaqus/CAE is associative (which means connected). Abaqus/CAE also has nonassociative (direct) capability, but that is not the focus of this paper.

Nonassociative translators (such as IGES, STEP, ACIS, and, Parasolid) are called from the geometry tool and a file is saved in the requested format. That file is then read into the FEA software. There is no connection or association between the model in the CAD software and the model in the FEA software.

But, Abaqus/CAE Release 6.6EF (Extended Functionality) introduced an interactive link, connecting FEA software to CAD software. This is an improvement over other companies with CAD Connection capability.

Prior to Release 6.6EF, if the user wanted to make a change to the CAD geometry, the translation process needed to be repeated for each geometry change. All FEA meshing, applying loads and boundary conditions, etc., would have to be repeated. Therefore, few design iterations with the CAD software were done. And in many cases the final CAD model was different than the final analysis model.

In Release 6.6EF of Abaqus/CAE, a feature was added to maintain a connection between Pro/E and Abaqus/CAE by using translation software from Elysium, Inc. The Elysium translator allows an associative connection between Pro/E and Abaqus/CAE. A link is maintained that allows geometry updates, while retaining all loads and boundary conditions on the surfaces. Regions that are made thicker retain their loads and boundary conditions. However, geometry changes do require updating the mesh. And mesh seeding will be retained on all uncut geometry edges.

This paper summarizes an evaluation of capabilities in the Associative Import plug-in. The focus is on Automatic Associative Import, but there will be mention of Manual Associative Import.

Import steps will be described, followed by the steps required to make updates. An example model will be used to show some of the capabilities. This will be followed by encounters and an overall summary.

2. Import Steps

Assuming that the Elysium translator is already set up in both Pro/E and Abaqus/CAE, one can proceed to the following steps.

1. Open Pro/E and read in the Part (component) or Assembly (group of components) to be imported.

2. With the geometry active in Pro/E, Abaqus/CAE is opened and the geometry is ready to be imported. Go to the Assembly module in Abaqus/CAE. Then select Tools/CAD Connection from the menubar as shown in Figure 1.

Figure 1. CAD Connection.

3. A dialog box will appear requesting a CAD Connection Port number (Figure 2). The port number is used by Pro/E to communicate with Abaqus/CAE during an Automatic Associative Import.

   [*This is the number found in file proe_abaqus.properties (dot properties file) on the line PORT=49178. The number, 49178, could be replaced by a user specified number. It must be a number from 1025 to 65535. The Auto-assign number is 49178, even if the PORT number was changed in the dot properties file. Therefore, if the user changes the number in the dot properties file, that number must be specified for an import connection. For example, if PORT=1025 was entered in the proe_abaqus.properties file, the user must select "Specify port" in the dialog and enter the number 1025. Selecting "Auto-Assign" or entering any other number will not work.*]

   Select the Enable button to accept the Auto-assigned port number (if the PORT number in file proe_abaqus.properties was changed see italics above). Selecting the Specify port toggle and entering a number different from that in the file will not work (assuming the PORT number is 49178). The figure shows the option of specifying a number.

Figure 2. CAD Connection Port.

Figure 3 shows the port number in the Abaqus/CAE information window. The default number (49178) is the number generated from Auto-assign. This is the number that is assigned when the software was installed (if the user did not change it to a different number in the range).

Figure 3. Auto-Assigned Port Number.

4. From the Pro/E menubar select Abaqus/Open in CAE (Figure 4).

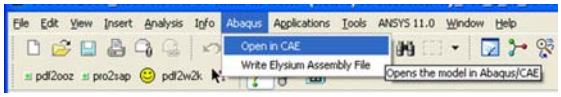

Figure 4. Open in CAE.

5. The view will be in the x-y plane, looking into z-axis, for the first import of an Assembly model. But the coordinate orientation of the model will be the same as in Pro/E. The Assembly in Figure 5 is an Exhaust Cylinder Section of a Gas Turbine and has been rotated from the x-y plane for a better view. The component color from Pro/E should also come into Abaqus/CAE. However, Figure 5 shows that there are some exceptions. The blue color did not import (the Cylinder Cover is blue in Pro/E and gray in Abaqus/CAE). Colors assigned at the Part level are imported in, while those assigned at the Assembly level are not.

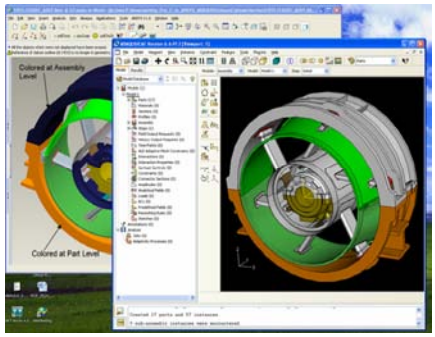

Figure 5. Assembly Model.

With the geometry in Abaqus/CAE, Pro/E can be closed while the analysis is performed. If there are immediate geometric iterations, Pro/E should be kept open.

3. Update Steps

Once the model is in Abaqus/CAE, updates are simple. Modifications trigger the regeneration process in Abaqus/CAE and can include suppressed features or unsuppressed features. Modifications can also include making changes to geometry. And it is not limited to Pro/E. The geometry can be modified in Abaqus/CAE also. Abaqus/CAE keeps track of adjustments within Abaqus/CAE, so that when an update to Pro/E is made, those adjustments, are modified as they relate to Pro/E updates. As an example of making a model update, the Cylinder component color will be changed in the Assembly Model shown in Figure 5.

Figure 6 shows the Cylinder component after the color was changed. The command in Pro/E Wildfire 2 is to pick View/Color and Appearance from the menubar. The Appearance Editor dialog box will appear. Select the color, then select the apply button. Now use the steps below as general updating steps.

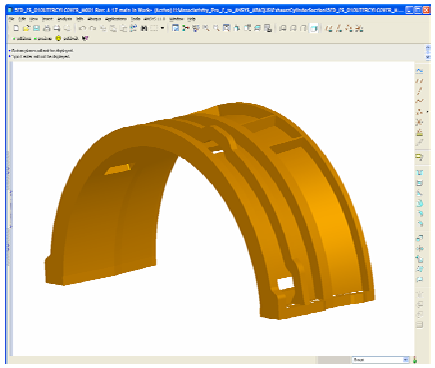

Figure 6. Cylinder Cover Model in Pro/E.

1 In Pro/E make the desired adjustment. [*In this case, the Exhaust Cylinder Section Assembly would be regenerated to capture the Cylinder Cover color change, by selecting Edit/Regenerate from the menubar. The color is now updated in the Pro/E Assembly Model.*]

2 From the Pro/E menubar select Abaqus/Open in CAE (as shown in Figure 4) to import the color change into Abaqus/CAE.

3 The update will be automatic. [*For Parts the user must toggle the Module dropdown (Figure 7) from Part to Assembly to see the change. This is because although the Assembly Module is active, the screen is blank because only the Part was modified. But the toggle described above alerts the Assembly to be updated by the Part.*]

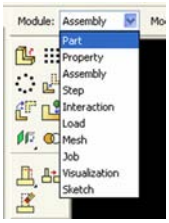

Figure 7. Module dropdown selection.

4 Figure 8 shows both the Pro/E and Abaqus/CAE models updated. Remember that for Parts it is not automatic.

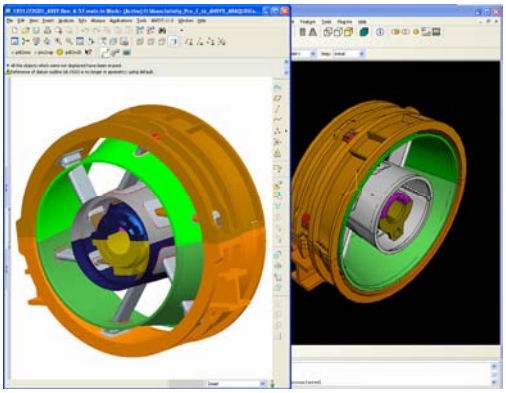

Figure 8. Cylinder Color Change.

4. Interface Example

A simple model (Figure 9) of two square blocks (designated as B1 and B2) will be used to demonstrate the most basic adjustments.

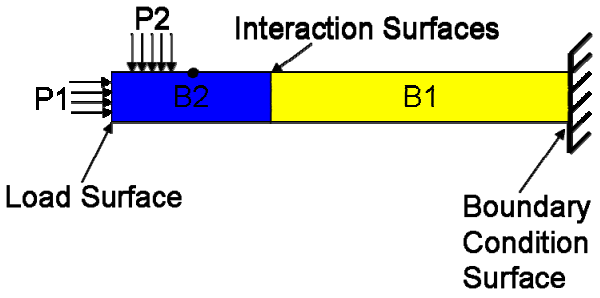

Figure 9. Dual Box Beam Model.

The model was set up in preparation for a change on different surfaces (a load surface, an interaction [set of surfaces] for contact, and a boundary condition surface for constraints). The changes include dimension changes and a hole through both blocks. A hole will determine if the surface conditions are lost due to the hole. P1 and P2 are pressure loads applied to Block 2. The top of Block 2 was split using a free form (tweak offset) feature in Pro/E. A fixed constraint was applied on the right end of Block 1. Figure 10 shows the dimension parameters.

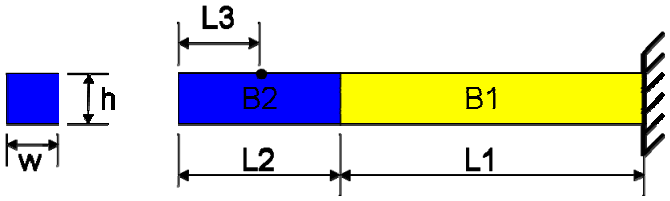

Figure 10. Dual Box Beam Dimension Parameters.

The split surface dimension L3 is 25 mm from the left surface end. All parameters of the example are listed in Table 1.

Table 1. Dual Box Beam Identifiers and Values.

Item	Parameter	Value

B1	Block 1	----
B2	Block 2	----
P1	Pressure 1	100 MPa
P2	Pressure 2	50 MPa
L1	Length of Block 2	100 mm
L2	Length of Block 1	50 mm
L3	Length of Split Surface	25 mm
h	Height	30 mm
w	Width	30 mm

Figure 11 shows the Pro/E model of the Dual Box Beam based on the dimensions in Table 1.

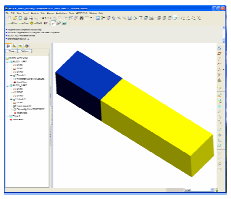

Figure 11. Dual Box Beam in Pro/E.

Figure 12 shows the part in Abaqus/CAE.

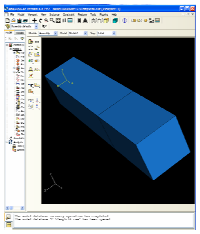

Figure 12. Dual Box Beam in Abaqus/CAE.

The loads, interactions, and boundary conditions (fixed end constraint) were added to the model. In the Abaqus/CAE Model analysis tree under the Assembly section, select STEP and create a

new Step. Create pressure loads P1 and P2 under Loads. Create a fixed constraint under the BC selection. Create a Tangential and Normal contact under Interactions. See the Abaqus/CAE documentation for steps to do this if more direction is needed. Figure 13 shows the loads and boundary conditions with no hole.

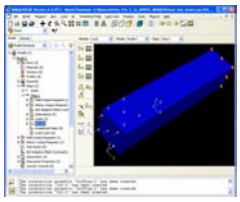

Figure 13. Dual Box Beam in Abaqus/CAE with Loads.

The first adjustment to the model is a hole through both blocks to determine if the loads, interactions, and constraints will be retained. The hole was created in Pro/E and was imported into Abaqus/CAE. Figure 14 shows the unsuppressed holes in Pro/E.

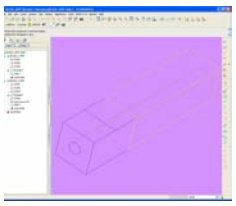

Figure 14. Dual Box Beam in Pro/E with Holes.

Figure 15 shows that the interaction (defined as a Set) is retained with the hole.

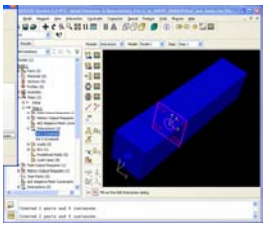

Figure 15. Dual Box Beam in Abaqus/CAE with Interactions.

Figure 16 shows that Pressure 1 (defined on a Surface) is retained with the hole.

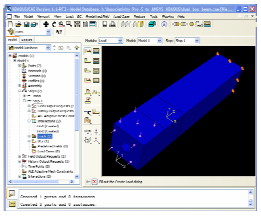

Figure 16. Dual Box Beam in Abaqus/CAE with Loaded Hole.

Figure 17 shows that the fixed end constraint boundary condition was retained with the hole.

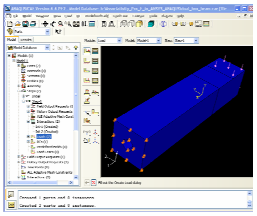

Figure 17. Dual Box Beam in Abaqus/CAE with Constraint.

The second adjustment was a height change on Block 2 from 30 mm to 15 mm. Figure 18 shows the adjustment in Pro/E.

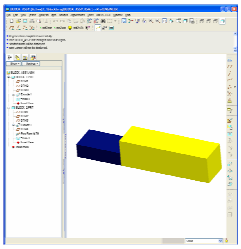

Figure 18. Block 2 Height Adjustment in Pro/E.

Figure 19 shows the adjustment in Abaqus/CAE. One can see that Pressures are retained.

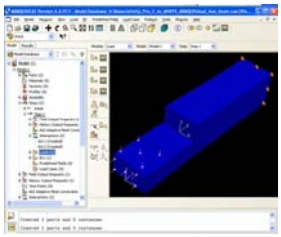

Figure 19. Block 2 Height Adjustment in Abaqus/CAE.

Figure 20 shows the results of a 15 mm width adjustment and again the Pressures are retained. Note the top split surface load was retained.

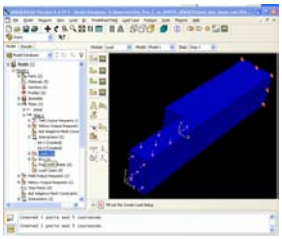

Figure 20. Block 2 Width Adjustment in Abaqus/CAE.

Figure 21 shows the result of a 1 mm radius added as an end round at the edge intersection of the Pressures. It is shown that the Pressures are retained on the existing surfaces. The user will need to add a Pressure on the round, if desired.

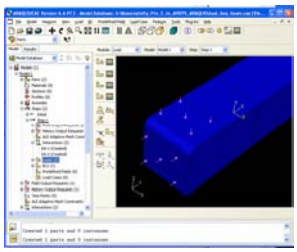

Figure 21. Block 2 Width Adjustment in Abaqus/CAE.

The above examples show that the loads, interactions, and boundary conditions are retained during typical model adjustments. Table 2 is a summary of import capability verified.

Table 2. Import Capability Verified in Abaqus/CAE.

#	Import Capability
1	Surface loads, interactions, and boundary conditions remain in place subject to a hole.
2	Surface loads and interactions remain in place subject to a thickness change (height or width).
3	Surface loads and interactions remain in place subject to split surfaces.
4	Surface loads and interactions remain in place subject to an added radius.

Be mindful that these adjustments were performed with clean geometry and straight forward adjustments. Some adjustments could raise questions and concerns. One such adjustment was in the surface split. The surface was split to apply the load on a section of Block 2, but if the feature is suppressed the load will not be there. The user needs to be sure that the outcome is what is desired when modifications are made.

5. Interface Encounters

While the majority of the effort was spent importing three-dimensional (3D) models, some time was spent trying to import two-dimensional (2D) models. It was later learned that there is no current Pro/E Associative Interface Support on 2D geometry from Abaqus. However, 2D geometry experiences will still be reviewed, since there are users that would like 2D geometry on rotors, rings, etc.

5.1 2D Encounters

Two encounters are worth noting for how import works for 2D geometry (Automatic Associative Import and Manual Associative Import.

5.1.1 2D Encounters: Automatic Associative Import

For Automatic Associative Import a rotor model was to be imported. Figure 22 shows the 2D model of the rotor section to be imported.

Figure 22. 2D Rotor.

The attempt was unsuccessful. Considering that it may have been a problem with the model, a new surface model was created in Pro/E. This attempt was also unsuccessful. The conclusion is that 2D geometry will not import using the Automatic Associative Import.

5.1.2 2D Encounters: Manual Associative Import

For the Manual Associative Import, colleagues from our Mülheim, Germany office did a study (Reference 3) on importing both 3D and 2D geometry. They concluded their study with four findings regarding 2D Manual Associative Import.

The first regarded coordinate systems. For axisymmetric models, Abaqus requires the geometry length to be along the y-axis. But a system model has the rotation axis defined by the z-axis. This is a problem because the geometry with all of the other Pro/E parts is a major driver. To manually convert the axis each time loses the affect of associative import. A request is to have the option of defining an output coordinate system dialog from the Pro/E side to maintain a relationship.

A second concern regards the merging of lines as they are imported in. The user divides lines in Pro/E for load division preparation. These lines are merged (by the import software) into a single line during the import to Abaqus/CAE. A request is for a control to leave the lines unmerged.

A third finding is models with multiple surfaces fail during the import, such as the rotor in Figure 22. The fix is to have the import allow a multi-surface part.

And the fourth finding is that free partitions work randomly.

It is hoped that 2D support will occur in the future.

5.2 3D Encounters

Two 3D encounters are worth discussing, the Dual Box Beam Example Model and a Single Blade Model. The encounter with the Dual Box Beam was unexpected. Other models will be tried to determine if the problem is repeatable. The Single Blade is noteworthy in determining how the software handles solids with surfaces in the model. So far it does not include them.

5.2.1 3D Encounters: Dual Box Beam

A noteworthy situation occurred with the Dual Box Beam model. There was a problem when suppressing and unsuppressing the hole and free form. After some trial and error, it was determined that having the hole defined in the assembly caused the problem (Figure 23). Specifically, for a freeform change only (suppressed or unsuppressed) in Pro/E, the import command does not update in Abaqus/CAE. Once a modification to the hole is made then the update activates. But only one of the holes updated. The face that the hole was defined on ended up being the block that would not change the hole. When the hole was defined on the other block, the problem switched ends.

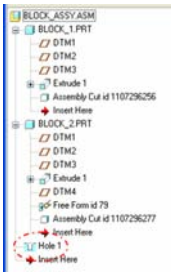

Figure 23. Model Tree with Hole at Assembly Level.

In Pro/E suppressing and unsuppressing the model worked without a problem. But Abaqus/CAE struggled with recognizing a change was made. When the hole was incorporated on each part, at the part level, everything worked fine in Abaqus/CAE (Figure 24). Unfortunately, this enlightenment happened after much trial and error. Other time consuming things were tried without success. Interestingly, after the holes were added in each part, the assembly in Pro/E did not have the colors in Abaqus/CAE, although the color was on the Part.

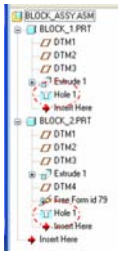

Figure 24. Model Tree with Hole at Part Level.

This could very well be a special problem. But it worked in Pro/E and no other way was found to make the model import better for this case.

5.2.2 3D Encounters: Surfaces with Solids

There will be times when a user will have surfaces along with solids in a model and would like to bring over all entities. There exist contact simulation problems where a ball will be dropped on a surface. It could be a surface of a plate with no thickness. The Single Blade model is not that type of problem, but it demonstrates the concern that surfaces are not importing. Figure 25 shows an assembly of a rotor segment and a LP blade. In Figure 25 the components are separated to show that they are both solids.

Figure 25. LP Rotor Segment and Single Blade.

Upon being read into Pro/E some features were lost by perhaps a tolerance difference. In the wireframe view of Figure 26 the red section in the center (the blade grooves) are surfaces with an opening somewhere, while the edge section is now one volume.

Figure 26. LP Rotor Segment and Single Blade Wireframe.

Figure 27 is the view imported in Abaqus/CAE.

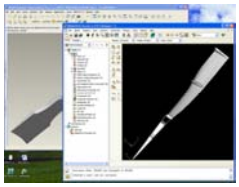

Figure 27. LP Rotor Segment and Single Blade Abaqus/CAE.

Figure 28 is a closer view a hole left from not including the surfaces.

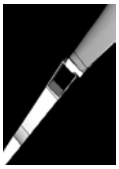

Figure 28. LP Rotor Segment and Single Blade Zoomed In.

Note the hole. The surfaces were not imported into the Abaqus/CAE environment. If the surfaces were imported in Abaqus/CAE, the Abaqus/CAE repair features could be used to bring the model back to solids.

6. Conclusion

Overall the tool is very good. There are areas for improvement, but users can be productive using this tool during design iterations. This document should be a good reference tool for new and existing users. The strategy of this paper was based on experience with using the tool. First, the steps to import a model were covered, followed by the steps to update a model. An example of known capabilities of the tool was described as a walkthrough. It was intentionally a simple example, so that users can try it and gain confidence in using the import tool. This saves users trial and error time. After becoming comfortable with using a tool, users are interested in what are known obstacles, so that time is not spent trying things that will not work. Table 3 shows pros and cons of the import feature to help in this.

Table 3. Import Pros and Cons.

#	PROS	CONS
1	User Friendly	No surface import capability (2D unsupported for Associative Import)
2	Loads remain on surface after surface is adjusted	No documentation to describe the capabilities
3	Constraints remain on surface after surface is adjusted	Not enough error messages when import does not
4	Loads on split surfaces remain after the surface is further adjusted	No material import capability
5	Inclusion of rounds does not modify loads on neighboring surface	No Bi-directional import capability
6	Allows multiple Pro/E windows to be open and can import the active one. The user can switch between either of them	
7	Stays connected if Pro/E is closed and re-opened	
8	Can run different models in Abaqus/CAE	

Readers can use Table 4 as a frequently ask questions section.

The bottom line is that in a production environment, reduced time in design iterations is the biggest return on investment. Once the model is read into Abaqus/CAE, the analyst applies loads and boundary conditions and solves. The solution is evaluated to determine if the model passes the company acceptance criteria. When the stress evaluation shows that the acceptance criteria is not met, a design change is performed. That design change is a wall made thicker, a hole cut, or a small fillet radius replaced by a larger radius.

After the geometry change is made, the analysis will need to be performed again to see if this design change will meet the acceptance criteria. Connectivity to CAD is critical to the product life cycle. That is, when the design iteration is done to a final geometry, the geometry that was used in the analysis must be represented by the CAD that drives the parts to be made.

In summary, the benefits of using the Associative Interface, is demonstrated in its first use. After using a direct translation approach for many years, it is refreshing to be able to perform design iterations quickly, and maintain clean geometry in both the CAD tool and the FEA tool.

7. References

1. ABAQUS Release 6.7 Documentation, "Pro/ENGINEER Associative Interface User's Guide", September 2007.
2. ANSYS Release 11.0 Documentation, "DesignModeler Help: Menus/File/Attach to Active CAD Geometry", January 2007.
3. Kloss, B., N. Lückemeyer, H. Claßen, and J. Roberts, "Feasibility Study of Pro/ENGINEER to Abaqus/CAE Manual Associative Import", Steam Turbine Engineering, Siemens Energy Sector, Mülheim, Germany, November-December 2007.
4. SIMULIA Visual Tips: Associative import from Pro/ENGINEER to Abaqus/CAE, Version 6.7-1 and later.

8. Appendix A - Questions and Answers

Table 4 shows questions that users of the import capability may ask.

Table 4. Questions and Answers.

Q#	Question	Answer
1	What are the steps to import a Pro/E Part or Assembly model in Abaqus/CAE using the Associative Interface?	-Open Pro/E & read in a Part or Assembly -Open Abaqus/CAE. -Select Assembly from the Module dropdown. -Select Tools/CAD Connection from the menubar. The Port number dialog will appear. -Pick OK to accept the (Auto-assign port) number 49178 or enter a value for the Port Number. -Select Abaqus/Open in CAE from the Pro/E

		menubar. The Part or Assembly model will be imported into Abaqus/CAE.
2	What are the steps to update the model after the first import occurred and a change was made in Pro/E?	-Go to Pro/E and make the changes desired. -Select Abaqus/Open in CAE from the menubar in Pro/E. The Assembly model will automatically update in Abaqus/CAE. *[TOGGLE PART: If the model is a Part it will not be recognized in the assembly automatically. -Toggle between Part and Assembly from the Module dropdown. –Select the Module dropdown arrow. –Select Part. – Select the Module dropdown arrow. –Select Assembly. The Part will now show with the change.]*
3	Can a different model be brought into Abaqus/CAE after the first import occurred?	Yes, but know the intent. -COMBINED: If the intent is to include the new model with the active model, make sure both models are in the same units, because the scale of the two will be drastically different otherwise. -VARIANT: If the intent is to have a variant of the Abaqus/CAE model, create a new Model environment (i.e. Model-2) and make it active. -NEW ANALYSIS: If the intent is to have a new analysis file, -Select File/New in Abaqus/CAE. Save the current model if desired. A new Abaqus/CAE environment will open. *Do the Following for either intent* - Select Abaqus/Open in CAE from the Menubar in Pro/E. (With desired model in Pro/E active.) Abaqus/CAE will automatically refresh. [see TOGGLE PART in Q2 if the new model is a Part]
4	Can any number be used as a Connection Port Number in Abaqus/CAE?	No. The acceptable number must be from 1025 to 65535. The Auto-assign number is 49718. This is the default number. It is good unless the number in the file is changed. If it is then that number must be used. -(User Assigned Number) Select Tools/CAD Connection from the Abaqus/CAE menubar. A dialog box will appear. Pick the Specify port toggle. Enter a value for the Port Number in the edit window. That number has to be the same number entered in the file proe_abaqus.properties on the line PORT=49178. The number replaced 49178. If changed and Auto-assign is selected when

		trying to import model from Pro/E a message in the display window will be “Either Abaqus/CAE is not started or wrong port number is specified.”
5	Can a different connection be started without exiting Abaqus/CAE?	Yes. If the user closed the active Pro/E file and opened a new one, the connection is still active. Follow the steps to import a model.
6	Can a surface only Part be imported in Abaqus/CAE using Associative Import?	No. The current Release does not support 2D import (3D only).
7	Can an Assembly with solids and surfaces (Parts with openings, such as slivers) be imported in Abaqus/CAE using Associative Import?	No. Only the solids come in.
8	Can a consistent import occur between the two software programs?	Yes.
9	What is the orientation of the import model in Abaqus/CAE?	The geometry on first import comes into the x-y plane with the z axis facing the user.
10	Does applied loads stay on geometry?	Yes.
11	Does applied loads stay on geometry that has been cut?	Yes, if the cut is a hole.
12	Does the model get automatically remeshed after an updated import from a geometry change?	No. Manual remeshing is necessary. Mesh seeding will be kept depending on the change.
13	Can it automatically update the Abaqus/CAE model when a change is made in Pro/E?	No. The link requires initiation by a selection from the Pro/E menubar. That triggers Abaqus/CAE to check the model for changes.
14	What happens if a Pro/E to Abaqus/CAE import Connection is started and then Pro/E is shut down? Will the connection still be available when Pro/E is opened again?	Yes. The active Part or Assembly can be imported in Abaqus/CAE.
15	Can multiple Pro/E windows be up and associated with multiple Abaqus/CAE models?	Yes, but the connection is linear. Multiple sessions of Pro/E can be open. Abaqus/CAE can have different Model windows active.
16	What if Abaqus/CAE is shut down after making a CAD Connection and lPro/E remains open? Will it remember the Pro/E connection?	No. A new connection will need to be initiated.
17	Will multiple coordinate systems get transferred?	No. Only the default coordinate system will be imported.
18	Is there a numbering system on parts or surfaces?	No. Names only.
19	Does unit system get transferred?	No. Abaqus/CAE is a dimensionless environment. Whatever units the model is in gets

		transferred. There is no updating to this.
20	Does material data get transferred?	No..
21	Will Abaqus/CAE bring in a model that was imported as a format other than Pro/E? For example, a STEP file, Parasolid, or Iges.	Yes. Open a Part or Assembly. Select the type (i.e. iges). Select the file and accept the type (Part or Assembly). The model will be activated. Select Abaqus/Open in CAE and the model will automatically import into Abaqus/CAE.
22	Are parameters transferred?	No.
23	Will the datum cut (partition) on a bolt still work if the bolt shank diameter is changed?	Yes.
24	Will part colors be updated?	Generally yes. But if the colors are defined in the assembly and not in the parts, it will not be seen when imported into Abaqus/CAE.
25	Is the Pro/E to Abaqus/CAE Associative Import bi-directional? If a change is made in Abaqus/CAE that needs to be sent to the Pro/E model from Abaqus/CAE, can it update the Pro/E model?	No.
26	If the thickness of a Part is modified, will the loads or boundary conditions be lost?	No.
27	If the surface of a Part is split, will loads or boundary conditions be lost?	No.
28	If a hole in a Part is created, will the loads or boundary conditions be lost?	No.
29	Will mesh seeds stay on models if a cut occurred?	Yes. It will try to compress the seed onto the remaining portion of the cut.
30	Will mesh seeds stay on models with thickness changes?	Yes.