Akselos

Known Issues and Limitations

Release 2026.10

Last updated: October, 2026
Classification: Public

Warranty Disclaimer: The Software and System are provided “as is” and “as available”. By using the Software and System, You acknowledge that they are only meant to be tools used to assist in operational decisions which shall be made by professional engineers skilled and trained in the use and interpretation of such results.

Akselos Modeler and Akselos Portal

No	Description	Register Date	Status	Mitigation	Fixed Version	Measurement Notes
1	**Hydrodynamic parameters** used for wave load cases are constant on one beam element. This may lead to mismatched results compared to other software. This especially happens for beam elements near sea water level. Measurement is under the process.	Oct 5, 2022	**Resolved**	Split beams at sea water level, table's elevations	Version 2023.04	The software can detect changes of marine growth distribution within one beam element now. This affects all wave-related loads (dead load, buoyancy, etc.)
2	Users cannot define **fatigue parameters for hotspot locations on solid components**	Oct 5, 2022	**Resolved**	None	Version 2023.04	Design fatigue factor (DFF), SN curve can be defined for each hotspots of fatigue components
3	**Fatigue result visualization for solid components** are not yet supported	Oct 5, 2022	**Resolved**	None	Version 2023.04	Fatigue damage result on solid elements can be displayed as similar as hotspots on shell elements
4	**UI features to build beam systems** (jackets) from scratch are limited. There should be features to import beam systems from other formats, basic UI to create/modify and delete beam elements.	Jun 3, 2022	**Acknowledged**	None		

5	**UI features to modify beam system are limited.** The following features should be improved: - Assigning/modifying beam cross-section to beam elements - Splitting/merging beam elements - Creating/assigning/modifying hydrodynamic table to beam elements - Beam/node selection	Jun 3, 2022	**Acknowledged**	None		
6	Selecting property on the left panel should highlight related beam elements on the graphic window	Jun 3, 2022	**Resolved**	None	Version 2025.04	Users can select material ID on the left panel and associated beam elements will be highlighted
7	Font size is small Widget sizes are not dynamically scaled	Jun 3, 2022	**Acknowledged**	None		
8	**Creating offset at 1D beam joint features** A UI to create offsets for braces at joints. Users can modify gaps between braces.	Jun 3, 2022	**Acknowledged**	None	Version 2023.04	Users are able to create rigid connectors which have the same axis with parent beam elements now. The rigid length is computed automatically based on cross-section of chord/leg (using outer diameter). More configuration options are under progress
9	**Load case creation workflow is complicated** There are several steps to create a simple load case via UI. This could be troublesome.	Jun 3, 2022	**Resolved**	None	Version 2024.01	
10	Configuration for pushover analysis is complicated	Jun 3, 2022	**Resolved**	None	Version 2023.05	Implemented many UI to configure Pushover analysis, including a wizard to set up analysis models for Risk-based Inspection.

11	Define load combination in load combination	Jun 3, 2022	Acknowledged	None		
12	Visualization of node/boundary ports/constraints in a more convenient way. I.e. different symbols for fully fixed, partial fixed, pin boundary conditions.	Jun 3, 2022	Resolved	None	Version 2023.11	-
13	Mapping function is not fully supported in terms on UI and solver functionalities	Mar 13, 2023	Resolved	None	Version 2023.08	-
14	Moving and rotating 2 components in sequence but the effect is applied for both components	Mar 20, 2023	Resolved	Save the model and restart Akselos Modeler	Version 2023.05	-
15	"Load More" feature takes much time on some particular models (e.g. Coker)	Apr 24, 2023	Resolved	None		
16	Token Framework Rev2 which can be applied to some customers since September 2023 has the following limitation: + Users cannot check remaining tokens of the organization + Users can only see token usage due to solving jobs	Sep 26, 2023	Resolved	Akselos Support can help providing the remaining tokens	Version 2024.01	-
17	Some parameters of Drag load cases (Morison loads) such as current_depth, current_velocity, etc. cannot be edited using UI	Sep 26, 2023	Resolved	Modify the information with AKS JSON format manually	Version 2023.11	-
18	Created beam components are stored in AKS folders by default. Model training is not yet supported for AKS folders.	Jan 15, 2024	Resolved	Move beam components from AKS folders to “components” folder of collections. There is an UI supporting this operation.	Version 2023.04	-
19	Akselos Modeler does not allow enter some expressions (red box border) even if the expression is correct in syntax	Jan 15, 2024	Resolved	Enter the expression manually via JSON Editor	Version 2024.04	-
20	The left panel does not change from “Model tree” to “Collection tree” automatically with “Open in Editor view” tool	Jan 15, 2024	Resolved	None	Version 2024.04	-

21	There is no warning / error when deleting Material ID / Cross-section ID that is being used in the model. A warning / error message is only introduced when the model is solved.	Apr 22, 2024	**Resolved**	None	Version 2024.10	There is an indication that the material ID is being used in the current model.
22	Cloning beam load cases with selected beam elements may create an unexpected relationship between 2 load cases: modify beam elements of one load case affect the other one.	Jul 15, 2024	**Resolved**	Do not clone those load cases, or do not modify applied beam elements after cloning.	Version 2024.10	-
23	**Token license server (AWS license server) may be overloaded during component training.** Users receive "Do you have a valid license key and enough tokens?" error message although there are still valid license and tokens. This only effects organizations with Innovate License Type only	Jan 8, 2025	**Acknowledged**	Resubmit training jobs again to resume training process Use "--one-job" training argument to combine training jobs		
24	The Portal's file upload progress bar stops at 99%	Oct 28, 2025	**Acknowledged**	If the progress bar is stuck at 99% for too long, reload the portal page (F5 / Refresh). The uploaded file(s) will then appear in the file list.		

Akselos Solver

No	Description	Register Date	Status	Mitigation	Fixed Version	Measurement Notes
1	Pushover solver and Boat impact are not yet supported for shell elements	Nov 2, 2022	**Resolved**	None	Version 2023.04	Pushover and boat impact including geometry nonlinearity and plasticity are not supported for shell elements with small strain behavior.
	Consider to use B-Bar for thermal expansion training We want to use B-bar for thermal expansion training by default because we are using B-Bar for FEA by default. At the moment, inconsistent integration types between FEA and training leads to large error indicators for some models.	Aug 19, 2022	**Resolved**	None	Version 2023.11	-
3	**Normal load defined by elem_xyz no longer worked** Error message returned when using elem_xyz for normal load: ---- A parameter was not assigned a value.(('o-normal_load_surface_200',) ('elem_xyz',)) ----	Oct 4, 2022	**Resolved**	None	Version 2023.11	-
4	**Enable Heat and Elasticity solves** Users need to have 2 separated collections for 2 different physics: heat and elasticity	Sep 30, 2022	**Resolved**	None	Version 2024.07	Support for FEA and RB-FEA analysis

5	**Maximum envelope solution for time-history simulation** This feature is not yet implemented for time-history solutions. It includes: - Maximum envelope Von Mises stresses to check with Allowable Stresses - Minimum envelope in-plane (panel) stresses to be used for shell buckling check	Sep 9, 2022	**Resolved**	None	Version 2023.08	Envelope solution for Von Mises stress is ready Envelope solution of minimum stress components in shell element's local coordinate system is available Strength check using envelope solution is not yet supported.
6	**Temperature dependent shells** is not yet supported	Nov 4, 2021	**Resolved**	None	Version 2024.04	-
7	**Nonlinear shells (NLGEOM and/or plasticity)** implementation is not finished yet. They are not fully validated.	Nov 4, 2022	**Resolved**	None	Version 2023.04	Pushover and boat impact including geometry nonlinearity and plasticity are not supported for shell elements with small strain behavior.
8	**Inertia Relief** implementation is not finished yet. It is still being improved and validated in the RB-FEA and Hybrid cases.	Nov 4, 2022	**Resolved**	None	Version 2023.11	-
9	**"Occlusions" in cavity radiation is not supported** Currently our ViewFactor matrix calculation ignores the fact that some elements may not be able to "see" other elements due to occlusions in the geometry of the cavity.	Jun 18, 2021	**Resolved**	None	Version 2024.07	-
10	Optimizations to reduce solve time and memory usage for large jacket models are ongoing	Oct 5, 2022	**Resolved**	None	Version 2023.08	Processing time and memory required for jacket simulations are reduced.

11	**Using Single Response** can push performance of multiple-time-step solve. However, there are some limitation: It only works for static elasticity projects (for Floating Wind, Fixed Offshore Jacket assets) The solving time is not displayed on the right panel of Akselos Modeler when it finishes	Apr 24, 2023	**Resolved**	Turn this option off	Version 2023.08	-
12	**The RB-FEA Solver for Thermal Physics** may experience limitations when working with temperature profiles that exhibit significant variations.	Jun 1, 2023	**Resolved**	When dealing with large temperature profiles, we strongly recommend exercising caution while using RB-FEA. It is therefore advisable to carefully assess and validate the results obtained from the software in such cases. In certain situations, the development of a Machine Learning (ML) model can be developed to anticipate stress levels under such circumstances. When used the detailed information regarding the ML specifications, testing methodology, and verification process will be provided to clients.	Version 2024.10	Using POD+GPR applet to perform simulations

13	**FEA Modal structural analysis** cannot generate more than 3000 mode shapes	Dec 1, 2023	**Acknowledged**	Users are able to generate higher mode shapes by entering the target frequency. Please contact Akselos Support for more information.		
14	Simulation result for unsymmetric cross-section for 1D beam element may be not correct	Jan 15, 2024	**Resolved**	None	Version 2024.04	-
15	**RB-FEA Modal analysis** introduces the following error message when solving a model with mapping functions --- |array_idx < A_theta_array[i].size() |Invalid index: 1, A_theta_array[i].size(): 1 Sending error: 10100 The job failed due to an unspecified error. Invalid index: 1, A_theta_array[i].size(): 1 ---	Jan 15, 2024	**Resolved**	None	Version 2024.10	-
16	**Some models with face port - node port connections** can have the following error during training: --- |map_find() error: key not found in file fe_systems/rb_fe/elasticity_condensing.C on line 617 Sending error: 10100 The job failed due to an unspecified error. map_find() error: key not found in file fe_systems/rb_fe/elasticity_condensing.C on line 617 ---	Jan 15, 2024	**Resolved**	None	Version 2024.04	-
17	Using both surface contact and geometric nonlinearity is not yet supported	Jan 15, 2024	**Acknowledged**	None		
18	Using both gasket element and geometric nonlinearity is not yet supported	Apr 22, 2024	**Resolved**	None	Version 2024.10	-
19	Using both gasket element and pre-tension is not yet	Apr 22, 2024	**Resolved**	None	Version 2024.04	-

	supported					
20	“Beam release” does not have effect when geometric nonlinearity is enabled	Oct 1, 2024	**Acknowledged**	None		
21	Usage of **springconnector3dofs, rotationalconnector6dofs, and springconnector6dofs** elements: “direction” option (which allows for a custom direction to be defined) is ignored when geometric nonlinearity is enabled (The axis of the element is used by default)	Dec 12, 2024	**Acknowledged**	None		
22	Usage of **rigidconnector3dof** element: When used as a "directional" constraint, the direction is not updated when using geometric nonlinearity with large rotations.	Dec 12, 2024	**Acknowledged**	None		
23	Plasticity and geometric nonlinearity are not yet supported for the **TRISHELL element type**	Dec 11, 2024	**Acknowledged**	None		
24	**Restart capability is not yet supported for** shell elements, rigidconnector6dofs elements, solvers other than the plasticity solver	Dec 13, 2024	**Acknowledged**	None		
25	**Calculation of Plastic Work** doesn't support gasket elements	Dec 14, 2024	**Acknowledged**	None		
26	**Calculation of Plastic Work** doesn't support Plasticity solver without geometry nonlinearity for shell elements	Dec 24, 2024	**Acknowledged**	None		
27	Rotating **beam-element components** creates incorrect results due to an inconsistency in coordinate systems in Modeler and the Solver.	Dec 25, 2024	**Resolved**	Avoid rotating the beam-element components in order to ensure that the beam element coordinate systems in Modeler and the Solver are consistent.	Version 2025.04	
28	**RB-FEA** solver doesn't support **B-Bar** with geometric mapping	Mar 6, 2025	**Acknowledged**	Use full integration option instead		

29	**Contact pressure** results are "not smooth" when using **non-conforming meshes**	Oct 1, 2024	**Resolved**	Use conforming mesh	Version 2026.01	
30	**Occlusion Detection in Thermal Cavity Radiation (AKA surface-to-surface radiation) is not yet fully supported.** In the case of cavity radiation, some surfaces may be occluded from others, which prevents radiative heat transfer between those surfaces. The occlusion effect is not yet fully accounted for in the case of the cavity radiation solver. The case where facets are facing "away from each other" is already omitted from the radiation calculations, but the case where there is an occlusion between facets is not yet accounted for.	Jun 12, 2025	**Resolved**		Version 2025.10	
31	“Multilinear isotropic hardening“ is not yet supported for 1D beam	Jun 23, 2025	**Acknowledged**			
32	**The vertical load is automatically converted to inertia in Modal Analysis:** - This is useful for jacket assessment, where equipment weights are usually modeled as vertical Z-loads. - However, in some cases, the vertical load does not represent the actual structural mass, which may lead to unexpected behavior.	Jun 23, 2025	**Acknowledged**			
34	**The distributed_point_mass_material operator is not equivalent to RBE3 in the modal solver :** - Operator with "MassAtRemotePoint" is not yet supported in Modal solver. - Operator with "MassAtTargets" is equivalent to applying small point masses to each target node individually.	Jun 23, 2025	**Acknowledged**			
35	**RB or Hybrid solves with substeps** do not yet support correct temperature interpolation for **temperature-dependent and varying temperature.**	Apr 28, 2025	**Resolved**		Version 2026.10	

36	**Mesh stitching** is slow during solving when the component includes **many subdomains**	Oct 16, 2025	**Resolved**		Version 2026.10	
37	Contact does not currently support cases where a single slave surface has multiple master surfaces.	Oct 24, 2025	**Acknowledged**			
38	Modal pre-stress implementation only supports linear elastic behavior with small displacements.	Jan 19, 2026	**Acknowledged**			
39	Reduced Integration for HEX8 is not supported yet.	Jun 11, 2026	**Resolved**		Version 2026.10	
40	Incomplete drilling stiffness term in the linear shell formulation	Oct 24, 2026	**Acknowledged**			
41	Incomplete drilling stiffness term in the default NLGEOM shell formulation	Oct 24, 2026	**Acknowledged**			
42	RBE3/RBE2 distributed load with NLGEOM is not supported yet.	Oct 24, 2026	**Acknowledged**			

Tools and Wizards

No	Description	Register Date	Status	Mitigation	Fixed Version	Measurement Notes
1	**SACS Importer does not have an option to add a pile-structure interaction (PSI) system directly** SACS Importer only reads SACS input files now. There is no option to consider PSI files during conversion. However, this can be done via the PSI plugin.	Jun 3, 2022	**Acknowledged**	Convert SACS model first, then create pile-soil system		

2	**Pile-structure interaction (PSI) support SAND and CLAY soil type only** PSI plugin only supports "SAND" and "CLAY" soil type from SACS input file. Other soil types are not yet supported.	Oct 5, 2022	**Acknowledged**	None		
3	**Performance of PSI tool is slow for some certain cases** Generating non-linear pile-soil systems takes longer than it needs, especially when the refined length is small (less than 0.5 m).	Oct 5, 2022	**Resolved**	Do not use refined length segment less than 0.5 m	Version 2023.04	Performance is improved: The non-linear pile-soil system can be generated much faster.
4	**The linear soil system generated by the Soil Linearization tool does not accurately represent the non-linear pile system.** For jacket structure, this tool generates linear stiffness matrices (6x6 matrix) at pile heads to replace the non-linear pile system for modal and spectral fatigue analyses. It ensures the behavior (displacements at pile heads) are similar between the non-linear model and the linear model.	Oct 4, 2024	**Acknowledged**	None		
5	**Enhancement to Soil linearization** **The current soil linearization approach has some limitations when dealing with high wave heights. A new algorithm is being developed to remove those limitations.**	Oct 1, 2024	**Acknowledged**	Using the current approach of soil linearization, double check the structural response before and after performing soil linearization.		
6	**SACS Importer does not support converting the phase fields with Length (L) and Time (T) units in the wave load.**	May 30, 2025	**Acknowledged**	The phase field needs to be modified after the model is converted.		

7	Importer incorrectly converts blocking factor in wave load cases	May 31, 2025	**Acknowledged**	Manually verify and adjust the blocking factor after importing the model.		
8	**Compute sectional Force and Moment** Results for second-order elements may be inaccurate in some cases	Apr 14, 2026	**Acknowledged**			
9	Beam System Building Creating a Beam System using "Insert Into Existing Group" may fail if the selected collection does not include CrossSection and Material Library.	Apr 14, 2026	**Acknowledged**	Ensure CrossSection and Material Library are included in the collection before using this option		
10	**Joint Creation tool** cannot generate mesh when using **Coreform 2025.8 and Coreform Cubit 2021** It generates 3D shell joints from 1D beam joints by calling Coreform Cubit in the background. There were changes from Coreform 2025.8 and Coreform Cubit 2021 that broke this tool.	Apr 14, 2026	**Acknowledged**	Use Coreform Cubit 2020 or Coreform Cubit 2023		

Assessments and Procedures

No	Description	Register Date	Status	Mitigation	Fixed Version	Measurement Notes
1	**Beam Codecheck supports one standard for one analysis** There is a demand on using a group of standards for beam codecheck, for example, API for tubular beams + AISC for non-tubular beams.	Jun 3, 2022	Acknowledged	None		
2	**Beam Codecheck returns UF=0 for invalid members** There should be no value (None/null) for those members	Jun 3, 2022	Acknowledged	None		
3	**Beam Codecheck runs analysis for the jacket and pile separately** They should run together as once and displays a whole visualization on graphic window	Jun 3, 2022	Resolved	None	Version 2023.04	Codecheck now can run both jacket model and pile at once
4	**Jacket Spectral Fatigue:** Stress concentration factor (SCF) computed for 1D beam joints using geometry only. SCF with the effect of loading is not yet supported.	Jun 3, 2022	Acknowledged	None		
5	**Jacket Spectral Fatigue:** only supports S-N curve with 2 slopes for now.	Jun 3, 2022	Resolved	None	Version 2023.04	Users now can specify both 1 slope or 2 slope SN curve from the user interface. They are able to define custom SN curves for each collection.
6	**Jacket Spectral Fatigue with time-domain stress RAOs** has slow performance as the jobs are not distributed well on computing nodes.	Jun 1, 2023	Resolved	None	Version 2024.01	-

7	**Jacket Spectral Fatigue with time-domain stress RAOs** only supports linear systems, and for beam elements.	Jun 1, 2023	**Acknowledged**	None		
8	**Shell Strength Check** implementation for DNV-RP-C201 and DNV-RP-C202 has the following limitations: + Lateral loaded pressure on panels are not yet taken into account + Stress components are averaged along panel edges + Stress of 1D stiffeners is not yet taken into account + 3D shell stiffeners are not yet supported	Sep 26, 2023	**Resolved**	None	Version 2024.10	-
9	**Wind Load on Floating Wind Tower** (generated from OrcaFlex data) hasn't considered tower fluctuation moment during simulation	Dec 1, 2023	**Acknowledged**	None		
10	**Current Load on Floating Wind using OCIMF** (generated from OrcaFlex data) hasn't been supported yet	Dec 1, 2023	**Acknowledged**	None		
11	**Second order wave load on Floating Wind** (generated from OrcaFlex data) hasn't been supported yet	Dec 1, 2023	**Acknowledged**	None		
12	**Stress concentration factor (SCF)** calculation returns only one set of values for 1 of 2 ends of the beam if it is connected directly to 2 different joints.	Jan 31, 2024	**Acknowledged**	Split that beam elements into 2 segments		
13	**Deterministic fatigue analysis** does not support beam components storing in AKS folders	Apr 22, 2024	**Acknowledged**	Move beam components from AKS folders to the "components" folder of collections. There is an UI supporting this operation.		
14	CodeCheck doesn't yet support beam elements with **offset parameters**		**Acknowledged**	Use a rigid connector instead of the offset parameter.		

15	AISC 360-16 Code-check is not yet supported for the **LbeamCrossSection**		**Acknowledged**			

While current testing and analysis have not revealed any known issues or limitations beyond what is listed on this document and current design considerations, we are committed to continuous improvement and will update this document if any are identified. This information is subject to change as we continue to develop and improve our software and system.

Should you encounter any issues or limitations not documented here, we encourage you to report them to our support team by emailing support@akselos.com. Your feedback is valuable in helping us ensure a high-quality experience.