

Workflow overview

Table of contents

1 Illustrative overview	1
2 Narrative overview	2

1 Illustrative overview

An overview of a Coreform IGA for Abaqus workflow is shown in the illustration below. In this illustration, processes are shown as rounded rectangles while files are shown as icons representing databases or text-files.

- A process that is colored **purple** signifies that it is a standard Abaqus process or workflow. A process that is colored **green** signifies that it is part of the Coreform IGA custom application or solver workflow. A process that is bi-colored **purple—green** signifies that the process can utilize both standard Abaqus and Coreform IGA workflows.
- A file-icon that is colored **purple** signifies that it is a file that would exist in a standard Abaqus workflow and users should be able to interact with these files as they normally would in a standard Abaqus workflow. A file-icon that is colored **green** signifies that the file's existence is unique to the Coreform IGA workflow and users should exercise caution before interacting with these files.

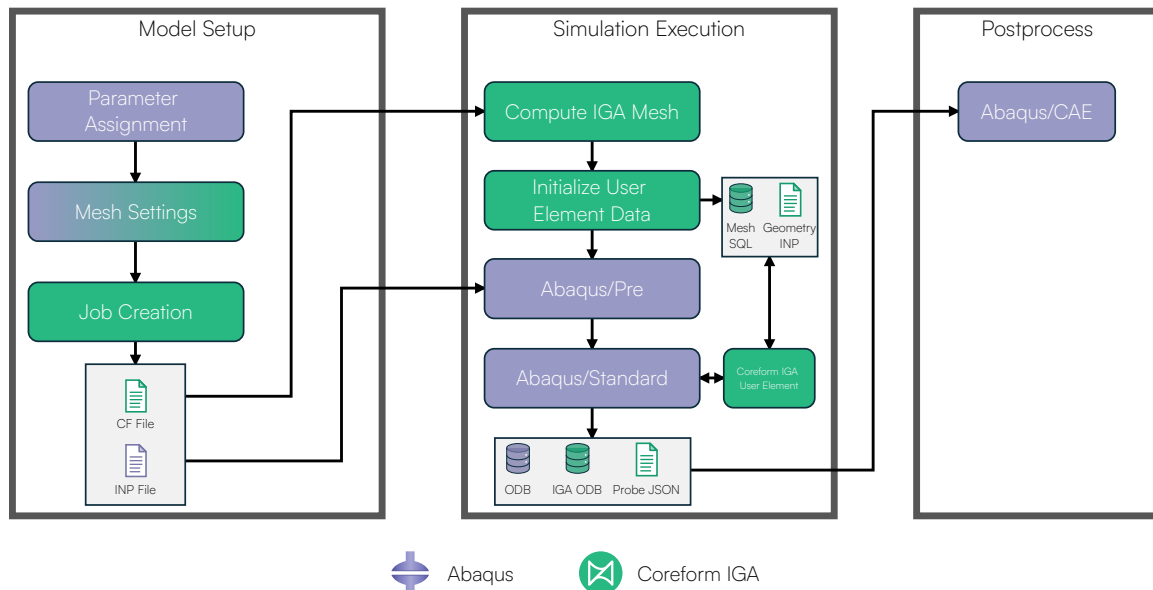


Figure 1: Overview of a Coreform IGA for Abaqus workflow.

2 Narrative overview

1. Model Setup *(Only Abaqus/CAE license required)*

No Coreform licenses are required for the model setup phase, only an Abaqus/CAE license is required.

- a. Parameter assignment
 - a. Traditional model setup in Abaqus/CAE, importing geometry, creating & assigning materials, defining steps & load conditions, etc.
- b. Mesh settings
 - a. Coreform IGA for Abaqus supports simulations comprising both regular Abaqus elements as well as Coreform IGA meshes. Users can use Abaqus/CAE's built-in meshing tools for regular Abaqus meshes, or the Coreform IGA mesh tool to define IGA meshes.
- c. Job creation
 - a. Simulations involving IGA meshes require the use of the Coreform IGA job tool to create and submit the job.
 - b. This step produces two files:
 - a. An Abaqus **INP** file whose name is `<job-name>.inp`
 - b. A Coreform **CF** file, whose name is `<job-name>.cf` that provides input for the trimmed-mesh generation routine

2. Simulation Execution

- a. Compute trimmed mesh *(Coreform IGA for Abaqus license required)*
 - a. Generates the trimmed spline mesh as defined by the user.
 - b. Computes element & basis function data (e.g., quadrature, extraction operators, extension operators, function splitting).
 - c. Writes out a **SQL** database file, whose name is `<job-name>.sql`, that contains the trimmed mesh and element/basis function data.
- b. Initialize user element data & translate Abaqus keywords *(Does not require a Coreform IGA for Abaqus license)*
 - a. Copies the original Abaqus setup deck, `<job-name>.inp`, to the protected source deck `<job-name>_source.inp`.
 - b. Reads `<job-name>_source.inp`, translates applicable Abaqus keywords into forms compatible with Coreform IGA, and writes the translated, runnable setup deck back to `<job-name>.inp`.
 - See the [Abaqus keyword support guide](#) for the translation behavior of each supported semantic case.
 - c. Converts element and basis function data into efficient data structures for Abaqus user-elements and stores this data in the same **SQL** database file, whose name is `<job-name>.sql`.

- d. Generates the IGA geometry deck `<job-name>_iga_geom.inp`, which contains `*USER ELEMENT` and related definitions. The translated `<job-name>.inp` references this geometry deck with an `*INCLUDE` keyword.
- c. Execute Abaqus/Pre (*Does not require a Coreform IGA for Abaqus license*)
 - a. Regular Abaqus/Pre execution
- d. Execute Abaqus/Standard with Coreform IGA User Elements (*Coreform IGA for Abaqus license required*)
 - a. Abaqus/Standard is executed with the `uniquelibs` option.
 - When submitted via Abaqus/CAE this option is automatically set.
 - If submitting a job via the command line, the user must specify `uniquelibs` manually in the command and make the Coreform user-element libraries in `<coreform-install-dir>/bin` discoverable to Abaqus/Standard. On **Windows**, add `<coreform-install-dir>/bin` to the `PATH` environment variable; on **Linux/macOS**, add it to `LD_LIBRARY_PATH` (equivalently, set `usub_lib_dir` in `abaqus_v6.env`). In `uniquelibs` mode either mechanism is sufficient to locate the `*_usub` libraries.
 - b. Abaqus/Standard loads shared libraries, `*_usub.[so|dll]` packaged by Coreform that provide the user element functionality.
 - If no Coreform IGA for Abaqus license is available the simulation will pause until a license becomes available.
 - c. Coreform IGA for Abaqus creates `<job-name>_iga_results.sql` at the start of the analysis. This results database is separate from the `<job-name>.sql` mesh and run-time-input database. It contains a copy of the IGA mesh together with the requested IGA result frames and fields written as the analysis proceeds.
 - d. Coreform IGA for Abaqus creates an ODB file with the name `<job-name>_iga.odb`.
 - Coreform IGA for Abaqus queries Abaqus/Standard for output field variables and stores them on a linear hex-tet mesh computed during Coreform IGA Mesh within this ODB file.
 - e. If the user specified a Coreform IGA for Abaqus *probe*, Coreform IGA computes the probe data and saves it to `probe_data.json`.
 - f. During execution, Abaqus will query Coreform IGA for Abaqus to provide linear system information (e.g., stiffness matrix, mass matrix, force vector).
 - g. After completion of the simulation, the Coreform IGA for Abaqus license is released.

3. Postprocessing

- a. Visualize and analyze results in Abaqus/CAE
 - a. The `<job-name>.odb` file contains any regular Abaqus elements defined by the user, as well as nodes from the Coreform IGA for Abaqus user-elements, as well as surface meshes generated by Coreform IGA Mesh that are used for boundary conditions, load conditions, constraints, contact, etc., and is compatible with Abaqus/CAE.

- b. The `<job-name>_iga.odb` file contains the linear hex-tet mesh generated by Coreform IGA Mesh and the (currently hard-coded to a limited set) output field variables.
- c. The `probe_data.json` file contains Coreform IGA for Abaqus probe variables.
 - a. While not natively supported by Abaqus/CAE, skilled analysts can use Abaqus/CAE's integrated Python console to `import json` and use the `json` module to extract the data as Python objects, and then use the Abaqus Python interface to convert to `XYData` objects for plotting in Abaqus/CAE.
 - b. Alternatively, the `json` file can be visualized in 3rd-party tools like Python (`matplotlib`), Julia, Matlab, etc.
- b. Visualize and analyze results in ParaView

Important

Alpha feature

Coreform IGA includes an alpha ParaView reader that can open `<job-name>_iga_results.sql` and visualize its IGA mesh and supported result fields. The results database itself is written by Coreform IGA during analysis; the alpha designation applies to the current ParaView reader and postprocessing workflow. Its interface, compatibility requirements, and supported result fields can change.