



Click2Extrude Metal 2018

Release Notes

General Enhancements

These enhancements apply to extrusion, tool deflection, and quenching tabs.

New File Extensions

The model files will be saved now with extension *.c2em, where c2em stands for "Click2Extrude Metal". The interface will continue to support opening old *.stmod and other geometry formats. Having a unique file extension will enable the user to double-click and load the model unambiguously into the correct application.

Job Manager Improvements

In this release, a new GUI is added to Create/Select remote services as a part of job submission dialogs. This GUI also has an advanced option to invoke the native Job Manager application from HyperWorks.

Start with Existing Mesh

The interface now supports creation of data decks with existing meshes. This will enable the user to manually create and use meshes for complex models. Meshes can be created in HyperMesh or any other mesh generator. At the time of job submission, the user can select this option, and Click2Extrude will complete the data deck by creating bearing profiles and all boundary conditions. Mesh provided by the user can be in Nastran BDF, HyperXtrude GRF, and OptiStruct FEM formats. In this release, starting from an existing mesh is not supported for coupled-OptiStruct analysis.

Localization of Material Property Dialog

Material property names are localized and translated to all supported languages.

Dynamic Unit Change in All Dialogs

When the user changes the unit system, those changes will be applied dynamically to all open dialogs.

Simulation Parameters Saved to Model File

All simulation parameters used for the analysis are stored in the model file. These values are restored both when the model is in session and when the model is loaded back in a new session. However, when the user starts a new model from CAD data, default values for process settings will be displayed. This enhancement is extended to other features such as billet creation.

Run Management Improvements

Run management in Click2Extrude is updated to closely follow the standards used by Inspire 2018. These include implementation of run history, automatic run folder creation, run status improvements to launch post-processing, and terminating the ongoing run from the run status dialog.

Model Orientation

Model orientation is improved to automatically align the model to +Z direction from any arbitrary orientation. This is achieved by using the "die exit surface" as the reference, and the user will have to select this surface to complete this action.

Extrusion

Edit Bearing Curve

A new feature is added to edit the bearing curve. Using this feature, bearing control points can be added or deleted along the curve.

Bearing Enhancements

Bearing control points extraction is improved to handle vertical/step bearing curves, achieving a smooth transition with new control points added on either side of the step vertical edge.

Coupled OptiStruct Simulation – Gap Contacts

The interface will now support creation of gap contacts (disconnected mesh) between two tool solids.

Tool Deflection

Combination of Mapped and User-Specified Loads

This feature will enable the user to combine mapped loads and user-specified loads. On selected surfaces, the user can specify loads manually and use this feature to override the mapped loads on those faces.

Temperature-Dependent Elasto-plastic Analysis

Click2Extrude is enhanced to perform temperature-dependent tool deflection analysis. This analysis will require temperature-dependent stress/strain data and will use the built-in data for this purpose. There is no GUI in this release to specify the required stress-strain material data via the interface, but the user can provide their own data. Click2Extrude exports this data as a text file in the run folder, and if the user provides a similar text file in the run folder, the interface will automatically use the user's data instead of its built-in data.

Contact Modeling Between Tool Solids

The interface is enhanced to handle contact condition between tool solids (as supported by OptiStruct solver). This has two major benefits: improving solution accuracy and speeding up meshing time.

Assigning Different Material to Tool Parts

The interface is enhanced to allow assignment of different materials to different tool parts.

Quenching

Support for All Quench Zone Types

The quenching interface is greatly improved to create all quench zone types supported by the solver. These include: spray cooling, immersion quenching, air nozzle cooling, and air fan cooling. The user can create and edit these quench zones with just a few mouse clicks. This is the major enhancement in this release.

Quench System Database

The quenching system database is now the central focus of quench system creation, and the workflow and icons are redesigned to support this. The main icon will now launch the quench system database where the user can load existing quench systems. The user can create new systems via the "Create" satellite icon and add them to the database. The user can also look at the summary from the table satellite icon.