AFDEX Newsletter Q1 / 2022

MFRC
www.afdex.com, mfrc@afdex.com
Tel: +82-55-755-7529, Fax: +82-55-761-7529

Contents

1. AFDEX_V21 Release

AFDEX_V21R02 was released at the end of October, 2021 and offers a new set of improvements in solver and Pre/Post-processors.

2. AFDEX_V21 Improvements

2.1 Gap Flow Control for 2D/3D Simulation

Following the increase in the case of the complete analysis considering die-structural-analysis and heat transfer analysis, the need of the direct control for gap flow between dies has been risen. The gap flow feature has been already provided in AFDEX_V19, and there has been continuous improvement in this feature. Now, in AFDEX_V21, sophisticated control is available for preventing flowing of a material into a gap by entering a clearance for each stage. Especially, in 2D, gap flow of a material gets through occasionally as shown in Figure 2.1(a). However, there was no way to control the gap flow between dies, although the clearance can be determined automatically during a simulation. By contrast, Figure 2.1(b) shows the simulation result with gap flow controlled by user, which was set 0.5mm for the clearance limit of gap flow. This feature is useful for the simulation that requires to prevent remeshing caused by flash and fluctuation of forming loads.

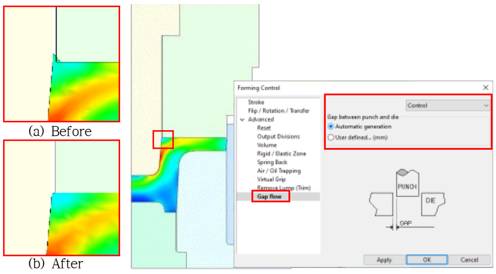


Figure 2.1 Controlling gap flow between dies

2.2 Improvement in Analysis using Binder

In AFDEX, the binder feature has been utilized for a force prescribed die including a blank holder. The implicit method algorithm implemented in AFDEX provides accurate solutions but has difficulty in imposing the given load through a die numerically. To resolve this problem, the penalty method modified specially is used in AFDEX. The penalty method stands for the method which allows penetration of a die into a workpiece considering the load imposed. Using the penalty method, it is possible that solutions that satisfies yield criterion does not exist if the high loads operate locally. This error and the inconvenience in controlling the motion of the binder are fixed in AFDEX_V21.

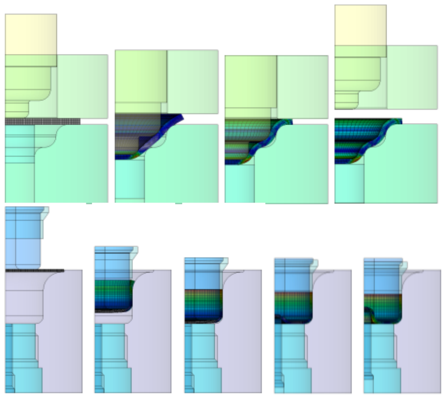


Figure 2.2 Analysis result with binder

2.3 3D Moment Save Feature

The need of multibody analysis technology is growing rapidly. Considering this trend, the graph of moment for both workpieces and dies is available in the newest version, whereas only the graph of moment for dies has been provided in the current version.

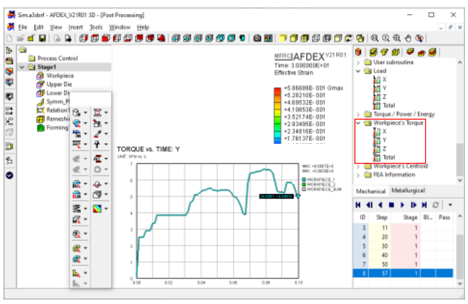


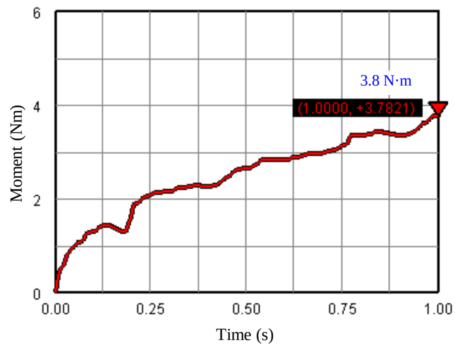
Figure 2.3 Graph of moment of a material

2.4 3D Sticky Die Feature

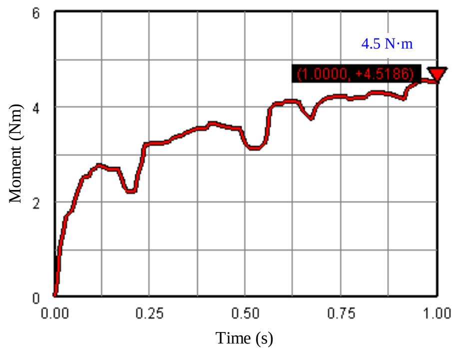
Recently, there has been development of computational speed-up in 3D simulation to solve a large problem with massive elements. However, increasing the number of elements cannot always be the best way for good simulation. Metal forming simulation is inherently exposed to the limit of numbers of nodes and elements. Half edge length of finite element in 3D increases eight times the number of elements and thus the global refinement of finite element mesh system cannot satisfy the researchers who need extremely refined mesh system.

This problem becomes more serious in case of multi-body simulation in metal forming. It is because the contact interface cannot be continuous in terms of slipping when the coarse surface contacts with die or material. It causes inaccuracy in dealing with friction and geometric constraint of preventing the slippage at the contact interface.

To cope with this matter, AFDEX_V21 provides the sticky die feature which adjusts the contact area at the interface with coarse finite elements. When the sticky die is employed, the triangle with one or two contact nodes at the interface between a material and a die or a material and a material can experience the frictional stress.



(a) Without sticky die



(b) With sticky die

Figure 2.4 Effect of sticky die on the torque

2.5 2D/3D Remeshing at Each Stage

The remeshing feature can be used for each stage in AFDEX_V21, whereas it has been used for only a project in the previous versions. The newly updated feature in the latest version enables full simulation run of continuous process without pausing.

2.6 3D Roll Forming Simulation

AFDEX_V20 has provided roll forming simulation using rigid-plastic finite element analysis. However, springback is one of the most important performance parameters in the roll forming simulation. Elastoplastic finite element analysis for the roll forming process is improved in the upcoming beta version.

The beta version of AFDEX for the roll forming process will be released in AFDEX_V21.

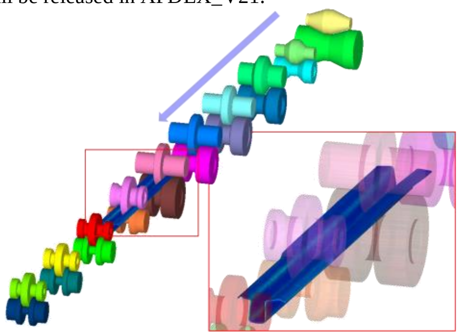


Figure 2.5 Roll forming analysis

In AFDEX_V21R01, the beta version of 3D roll forming is available. Although the roll forming simulation can be conducted by the original module, there has been some complexities to set special conditions. Now, one can control the boundary conditions between rollers before and after the forming. The new module will be released for licensed users.

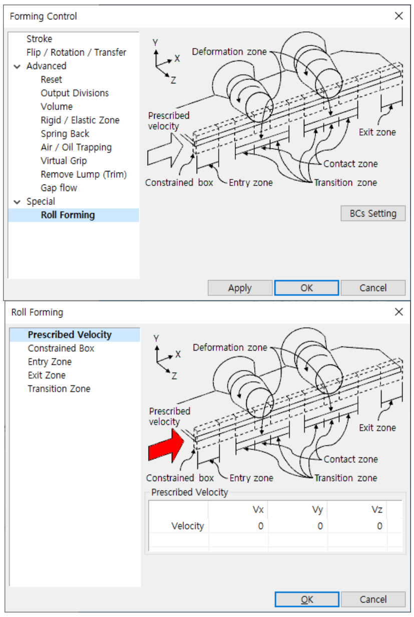


Figure 2.6 UI for controlling 3D roll forming analysis

2.7 3D Shearing Analysis

Shearing analysis is aimed to analyze the shearing process of a rod. Therefore, this feature might not operate well when it was used for the other processes including piercing and trimming in plate forging analysis. In AFDEX_V21, an improved function of shearing, piercing, trimming, etc. is given, which avoids the error mentioned above. On the contrary, the aforementioned problems could be solved using AFDEX's unique function for piercing and trimming.

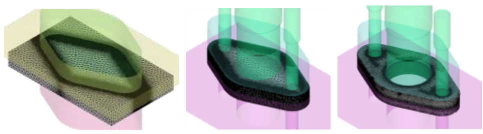


Figure 2.7 Simulation for shearing process in sheet metal

2.8 Multiple Damage Model in a Single Project

Damage is one of the important factors in forming processes, and some process requires to simulate the evolution of the damage with various damage models. For the convenience of users who perform a damage simulation, AFDEX_V21 provides multiple damage calculations with various damage models during single run. Currently, at maximum three damage calculations are allowed, and its results can be checked through the post-processor. The results of damage models can be compared a lot easier by using this feature.

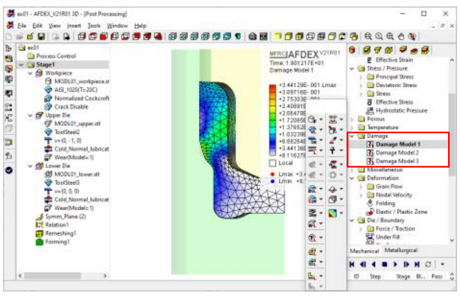


Figure 2.8 Post-processor with the results of multiple damage models

2.9 Quantified Brittle Fracture of Ductile Material

There are some cases that cannot be explained by the universal theory of ductile fracture caused by damage accumulation. The shearing analysis is one of them. For ductile materials, it is hardly possible to find fracture trace on sheared edge which is the trace of damage accumulation. A different example of brittle fracture of the ductile materials was found from the cold shell nosing process. We have suggested the concept of plastic deformation induced embrittlement to explain such fracture (Materials, Mar. 2021). The tensile strength of a material drops due to Bauschinger effect after the compressive deformation, which can result in brittle fracture depending on the material properties and level of plastic deformation. When tension is applied on the material whose yield strength became lower and embrittlement was grown, the material exhibits brittle fracture in case that the tensile stress greater than the reduced allowable tensile stress exerts. In this case, this brittle fracture occurs on the plane perpendicular to the axis of relative maximum tensile stress to the reduced allowable tensile stress. The fracture in Fig. 2.6 could not be predicted by any theory of ductile fracture while the suggested fracture model could predict the brittle fracture, as can be shown in the figure.

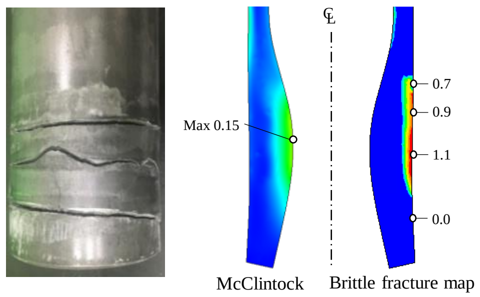


Figure 2.9 Brittle fracture of forgeable materials during cold shell nosing process.

2.10 Forming Instability Index

The use of the forming instability index was already suggested in the previous study (Int. J. Mech. Sci., Apr. 2021). The plastic deformation instability index provides an estimate of the instability of a material affected by strain, strain rate and temperature softening during forming. The index was formulated as follows:

$$\chi = \frac{\bar{\sigma}\dot{\bar{\varepsilon}}}{C_{in}} \frac{D\bar{\sigma}}{D\bar{\varepsilon}}$$

Figure 2.10 shows the change in instability index induced by dynamic strain aging effect during upsetting process.

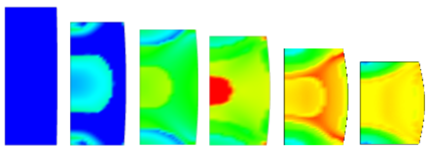
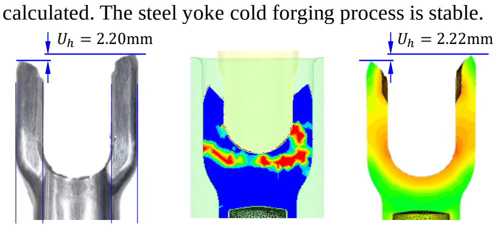


Figure 2.10 Change in instability index

Figure 2.11 shows a problem of ear height difference occurring in an aluminum yoke cold forging process, which can be predicted by FE simulation with an emphasis on the instability index. In this case, as shown in the Figure 2.11(b), the instability index shows the sign of the plastic deformation instability. In case of steel yoke forging process, such instability index cannot be calculated. The steel yoke cold forging process is stable.



(a) Experiment (b) Instability index (c) Simulation

Figure 2.11 Instability index of Aluminum cold forging for yoke

2.11 Force Control in Shaft Clinching

In AFDEX_V21, elastoplastic finite element analysis can improve the simulation of shaft clinching or rotary forming. The major concern of this simulation is the cavity between the shoulder of bearing inner race and the bent shaft because it is mechanically important in assembling and service. The research results suggest that force controlled forming die, that is, the rotary die which is same to the real process must be used for the prediction on the cavity region. Thus, the exact value of stress applied on an inner race can be obtained and predict the local plastic deformation occurring during homogenizing in the rear part of the process. Figure 2.12 shows the residual stress result predicted by the new beta features.

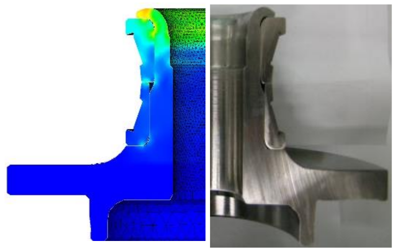


Figure 2.12 Shaft clinching simulation result

2.12 Bending Analysis with Flexible Mandrel

A feature for the bending analysis of a pipe with flexible mandrel which is one of the multi-body analysis applications is added in the release version. The main point of this application is treating a mandrel as an analyzing part of material. This scheme is an application of multi-body analysis functions, implying that they can be applied to various special forming processes in a creative manner.

Figure 2.13 illustrates the result of the bending analysis of a pipe with flexible mandrel using the beta feature which will be provided in AFDEX_V21. The multi-body analysis can be applied in not only this case, but also the other process similar to it.

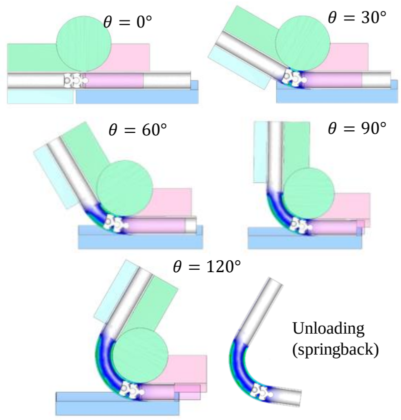


Figure 2.13 Predictions of pipe bending process with flexible mandrel

2.13 Flow Stress Models at Room Temperature

AFDEX_MAT has been used for obtaining the flow stress using MFRC's model K(ε)-n, which predicts result of tensile test exactly. AFDEX_V21 provides the new feature that can obtain the material constants with respect to various types of the flow stress model with using the tensile test data.

Especially, the flow stress predicted by AFDEX_MAT based on K(ε)-n model is highly accurate until a fracture occurs so that it can be used as a reference flow stress. Figure 2.14 is showing the evaluated flow stresses (Ludwik, Voce, Swift and Hollomon law) in terms of tensile test and compares to each other.

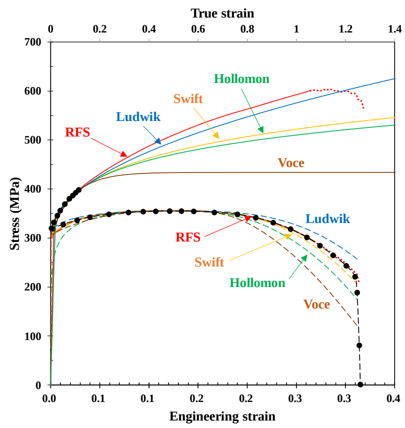


Figure 2.14 True stress-strain curves (upper) and their corresponding tensile test predictions (lower)

2.14 Flow Stress Models at Elevated Temperature

A variety of numerical models express high-temperature constitutive model of a material as a state variable. AFDEX_V21 provides useful flow stress models which can be used both in academy and industries.

Figure 2.15 shows the predicted flow stress models for Magnesium alloy, AZ80. The flow stress models such as the hyperbolic sine Arrhenius equation, the modified Hensel-Spittel model and the modified Johnson-Cook model are used based on the results of R. Ebrahimi et al. study. Ebrahimi, C-m and PLF models, on the other hand, are tested by AFDEX researchers. This comparison tells us how important the flow stress model is.

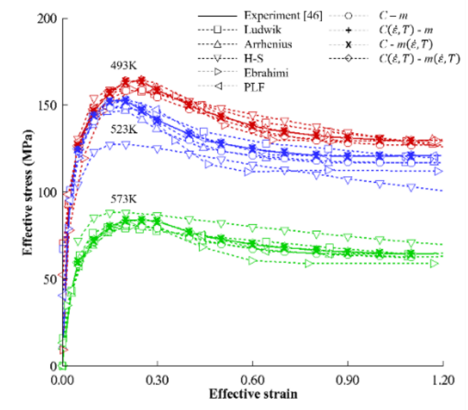


Figure 2.15 Comparison of high-temperature constitutive models for AZ80 (strain rate: 0.01/s)

2.15 Cladding Extrusion Process at Front End

For the futuristic technology, various activities for effective usage of materials and recirculation have been being made. For example, special or expensive materials have been purposely locally clad and the heat treatment has been applied partially to the mechanical parts for improving their service life. Figure 2.16 shows the predictions of plastic deformation and effective strain occurring during cladding of the front end of an extrusion with sheet material.

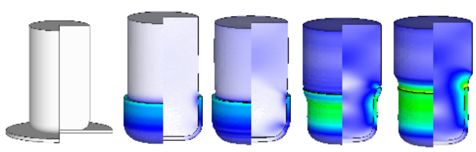
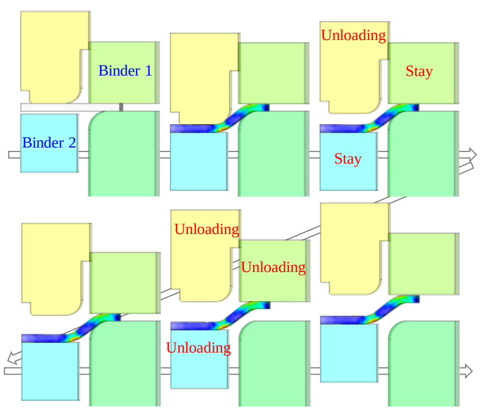


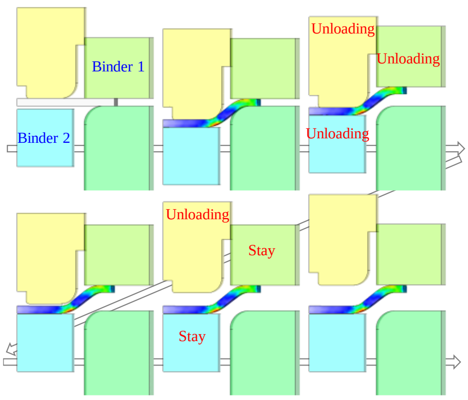
Figure 2.16 Analysis result of cladding extrusion process at a front end

2.16 Motion Analysis of Die Unloading

Previously, for the case without binders or dependent (slave) dies, backward motion can be simulated with information of velocity profile of dies. AFDEX_V21R02 provides the forming analysis according to the backward motion of the dies regardless of the binders and slave dies. The motion of the dies will be differed according to the upper and lower binder force which is entered as an input. Using this feature, AFDEX can simulate the whole process of the dies' returning to the starting position during plate forging. Figure 2.17 shows the examples of the motion analysis of die unloading.



(a) Case where binder force1 is greater than binder force2

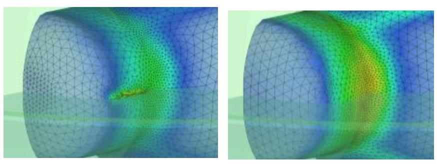


(b) Case where binder force1 is lower than binder force2

Figure 2.17 Unloading motion according to variation of the binder force

2.17 Gap Flow Control

Previous versions had a feature controlling gap flow occurring in the vertical direction but did not offer for the case of vertically-aligned dies, which the direction of the gap flow is horizontal. As shown in Figure 2.18(a), burr can be generated between the two actual dies, especially during enclosed-die forging. However, this can cause remeshing and increase in execution time, which is not proper in terms of obtaining a solution for the conceptual design. The new version allows to control all the gap flow according to what users enter for the input data.

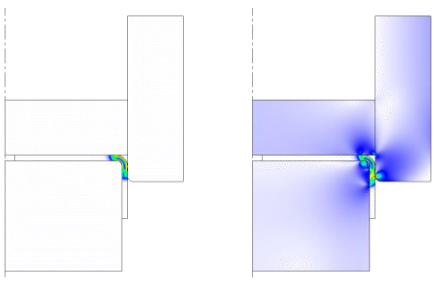


(a) Without function (b) With function

Figure 2.18 Controlling burr in closed-die forging

2.18 Die Structural Analysis of Plane Strain

Die structural analysis (DSA) can be categorized into a single die DSA and assembled dies DSA, and DSA of 2D plain strain problem for the assembled dies was not provided in the previous version. Now, AFDEX_V21R02 adds a feature DSA of 2D plain strain. The analysis for the assembled dies is widely used for the analysis considering the elastic deformation of dies. Figure 2.19 compares the results of effective stress after performing DSA of plane strain in the prevision version and AFDEX_V21R02.



(a) Original (b) Improvement

Figure 2.19 Results of DSA of plane strain (Effective stress)

2.19 Addition of CFRP Constitutive Model

The constitutive model of CFRP proposed by Wang et al. (Polym. Compos., 2002, Vol. 23, pp. 858-871) is added for analysis of CFRP. The input variables for the CFRP analysis are max/min flow stress, max/min effective strain, max/min temperature, Young's modulus and stress coefficient as a function of temperature.

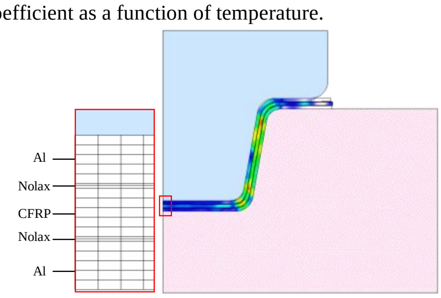


Figure 2.20 Application of CFRP constitutive model

This model is added in the constitutive library provided in AFDEX_SP of AFDEX_V21R02.

3. AFDEX_V21 GUI Improvements

3.1 Import DXF File

The error of automatic numbering dies in a DXF file occurring when there is large number of parts (assembly die) has been fixed. Also, the import DXF file dialog box can be used to change the position of upper/lower dies and to move stages. In the multi-body simulation, multiple workpieces can be set automatically.

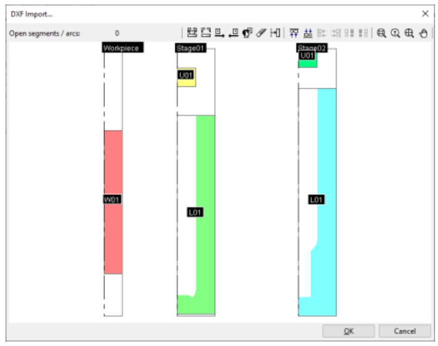


Figure 3.1 New import DXF file dialog box

3.2 Useful Features for Post-processing

AFDEX_V21R01 includes new features that are useful for post-processing. The features are as follows: Visualizing the centroid of a workpiece (Figure 3.2), dimensioning (Figure 3.3) and result probe (Figure 3.4) on the post-processor in 3D analysis.

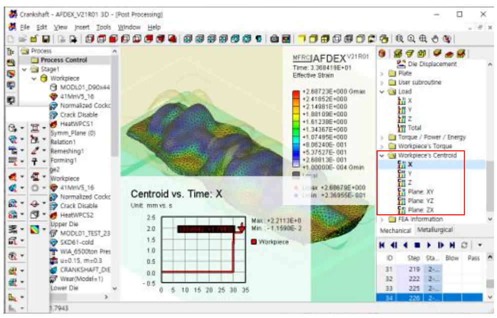


Figure 3.2 Visualization of centroid of a workpiece

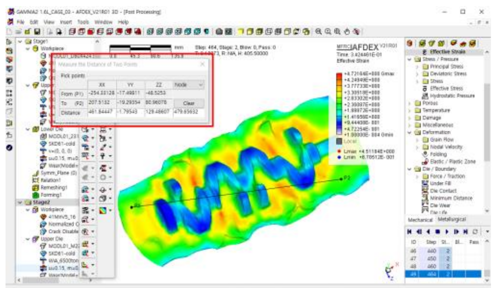


Figure 3.3 Dimensioning

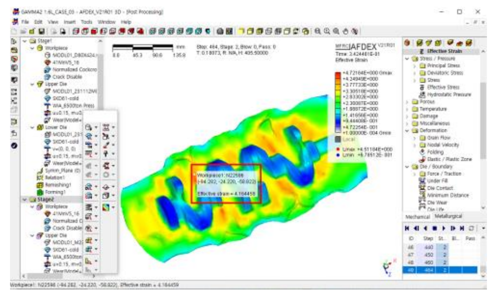


Figure 3.4 Result probe using Probe-at icon

3.3 Remeshing at Each Stage

AFDEX has been providing the multi-stage process analysis for 25 years, which controls condition of remeshing for whole processes. From the newest version, however, the remeshing control is able to set for each stage to distinguish the stage where needs the remeshing. For example, this feature can be properly used in the combined process analysis of the clinching process and the joint strength analysis.

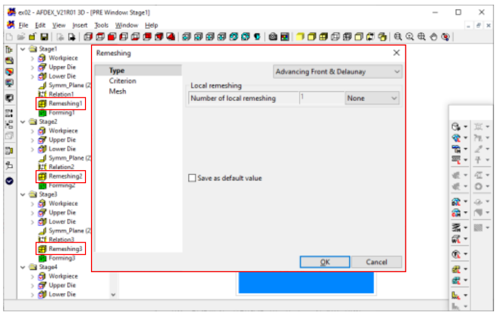


Figure 3.5 Dialog box for setting remeshing condition

3.4 Automatic Step Size Control and Mesh Generation

The dialog for controlling calculation speed and accuracy of an analysis is newly added in AFDEX_V21R01. There are two ways in which user can determine them by controlling the number of step size and mesh automatically or manually. The automatic setting will automatically determine the values of step size and mesh considering the forming stroke and the shape of workpieces and tools. For the case of using ‘User defined,’ one can enter the numbers of mesh and step size to determine the condition of any process.

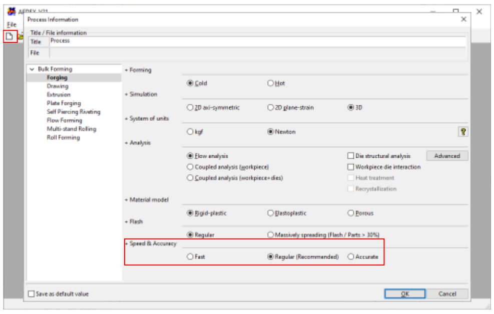


Figure 3.6 Dialog box of process information

3.5 Legend Bar Enhancements

In AFDEX_V21R01, one can change the size of the legend bar to optimize the results by clicking local/global button. In the previous versions, the options for changing the size of the legend bar were located inside the setting window.

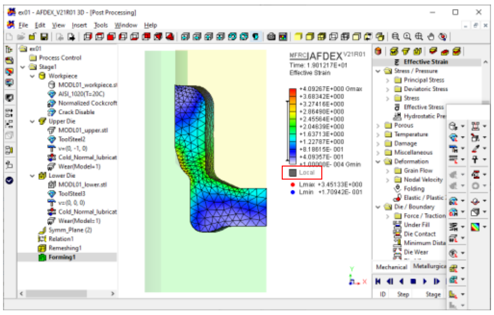


Figure 3.7 Changing the size of the legend bar

3.6 Coordinate Transformation

In the older versions of AFDEX, results were displayed on only Cartesian coordinate system. However, In AFDEX_V21R01 provides the transformation between two coordinate systems: Cartesian/Cylindrical coordinate system. Now, one can check the result of the state variables selectively.

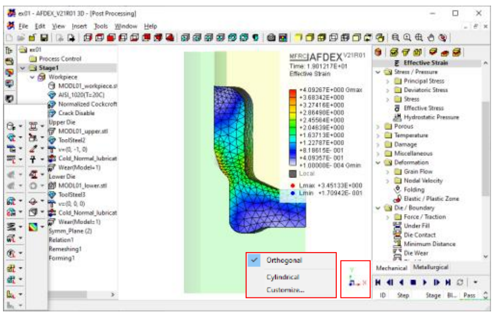


Figure 3.8 Coordinate transformation

3.7 Improved Dialog for Inputting Coefficient of Friction and Heat Transfer of Dies

Although this feature was implemented in the solver of previous versions, the dialog for the feature had not been provided in the pre-processor. Therefore, only users who had trained could run the simulation without the dialog. From the latest version, the friction behavior can be easily determined as a function of temperature, pressure and strain, while the coefficient of heat transfer for dies can be defined as a function of temperature and pressure.

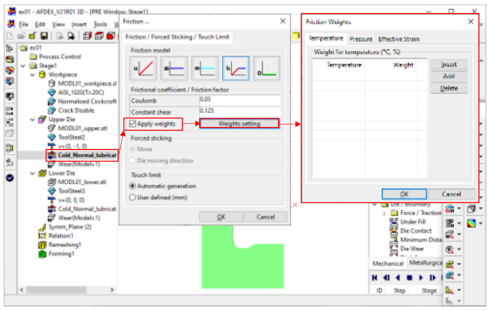


Figure 3.9 Dialog box for inputting the condition of friction

3.8 2D Automated Process Setting for Workpieces / Dies model

In cases of AFDEX 2D module, automated process setting is applied for each stage after importing DXF file which includes geometries information of material and dies. AFDEX_V21R02 can modify information of geometries of dies of an arbitrary stage, which previously was impossible.

3.9 3D Automated Process Setting for Workpieces / Dies Model

Previously, only AFDEX 2D has supported the feature of making overall processes automatically when importing DXF file. However, the new release supports the feature for both 2D and 3D, which STL file can also be utilized. For this feature in 3D, the automation of the process will be determined by filenames.

- Workpiece: **S**ii**W**jj, ii and jj stand for the stage ID number and the ID number of a workpiece, respectively.
  Ex) Stage: 1, workpiece ID number: 1, then S01W01
- Upper die: **S**ii**U**jj, ii and jj stand for the stage ID number and the ID number of an upper die, respectively.
  Ex) Stage: 1, upper die ID number: 1, then S01U01
- Lower die: **S**ii**U**jj, ii and jj stand for the stage ID number and the ID number of a lower die, respectively.
  Ex) Stage: 1, lower die ID number: 1, then S01L01

Note) If the filename includes any keywords described above, this feature will be applied.

3.10 AFDEX_SP new features and improvement

3.10.1 Point tracking

AFDEX_V21R02 adds an improved option in AFDEX_SP to view the point tracking history. One can plot a state variable such as the effective strain at a point tracked by entering a node No. or selecting an arbitrary point. Figure 3.10 shows a part of point tracking history.

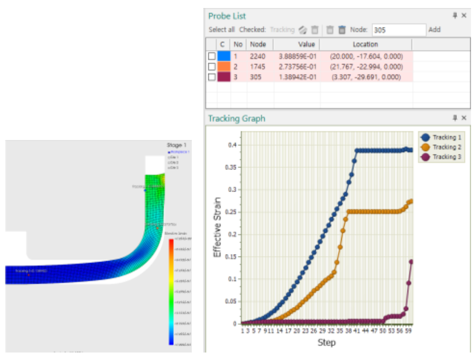


Figure 3.10 Point tracking history

3.10.2 Importing Mesh from NASTRAN format

AFDEX_V21R02 can import a geometry file which is in NASTRAN format (bdf file), and it is able to generate CTETRA elements for an analysis. Figure 3.11 shows the meshed model with bdf data.

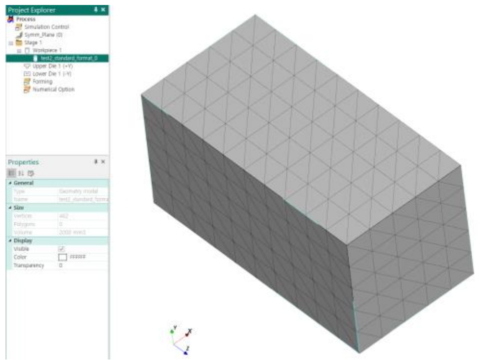


Figure 3.11 Meshed sample model with bdf data

3.10.3 FLC Input Data / FLD Output Data

AFDEX_SP provides a tool for entering input data of FLC (Forming Limit Curve) which Keeler’s equation is used. As shown in Figure 3.12, major strains and minor strains are plotted on a forming limit diagram (FLD), and contour values are described on the surface of the model.

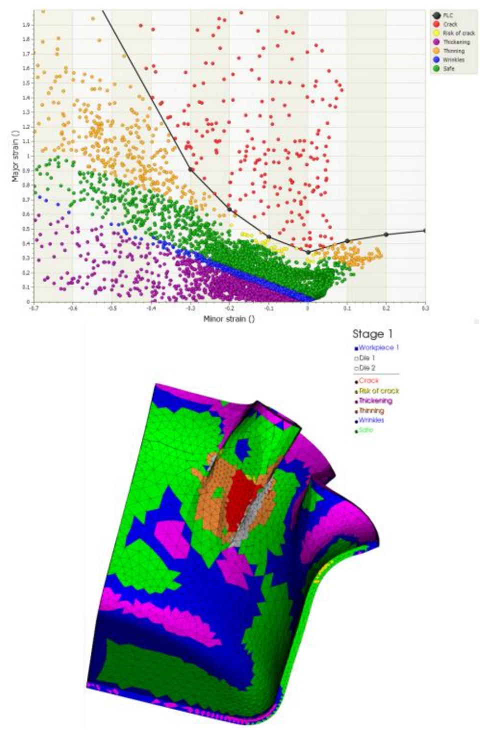


Figure 3.12 Formability with FLD (Forming Limit Diagram)

4. Notice

4.1 Altair manufacturing webinar

MFRC participated Altair Manufacturing Webinar Series 2021 held on June 22th, 2021. Dr. Mansoo Joun presented an online webinar entitled, “Perform Accurate Finite Element Modelling of Metal Forming Processes” for global users. Please find the recorded webinar video from the following link.
(https://www.afdex.com/archive/activities/19)

4.2 ICTP 2021

ICTP 2021 was held by Ohio State University, USA from July 25 to 30 (EDT). MFRC and GNU(Gyeongsang National University) attended this online event by presenting two papers. (https://www.tms.org/ICTP2021)

4.3 MSAM 2021

The 4th International Conference on Material Strength and Applied Mechanics (MSAM) was held on 16-19 August online and offline in Macau, China. In this conference, AFDEX development team presented four papers using AFDEX. Dr. Mansoo Joun was invited to this event to give a presentation of his research about material models for cold forging. For more details, please refer to the following link. (http://www.msamconf.org/Program)

4.4 ATCx Mexico 2021

MFRC attended a virtual ATCx Mexico 2021 and presented the webinar entitled, "Metal Forming Simulation using AFDEX." This event was held on June 2-4 and all the webinars were presented for Altair users and visitors in Mexico.

4.5 APA webinar in Japan

MFRC attended APA Japan webinar series and presented the online webinar entitled, "Metal forming simulation using AFDEX" for Japanese users. The webinar was held on April 20, 2021 and was focused on the following topics: Automatic simulation of multi-stage forming process, Multi-body forming process, Fatigue life prediction of a die and optimal process design & material properties, etc. For more details, please refer to the following link.

(https://www.altairjp.co.jp/resource/jp-apa-afdex)

4.6 SIAT 2021 EXPO in India

MFRC attended a biannual event in India, SIAT 2021 (Symposium on International Automotive Technology) and exhibition held by Automotive Research Association of India (ARAI) from September 29th to October 1st. In light of the ongoing global COVID-19 pandemic situation, SIAT 2021 was held as an online event.

4.7 IS-KSTP30

International Symposium on Technology of Plasticity (IS-KSTP30) for Celebrating KSTP's 30 Years Anniversary was held in Park Hyatt, Busan from November 24th to 26th. IS-KSTP30 served as a forum for exchange of ideas and brainstorming for the metal forming industry with participation of experts in various metal forming areas. MFRC presented the applications using CAE technique and had a Q&A session to interact with AFDEX users at MFRC's booth.

4.8 JSOL CAE Forum 2021 Online

JSOL CAE Forum 2021 was held online for 4 days from November 30th to December 3rd. This event introduced the latest technologies of various CAE packages provided by JSOL's engineering technology division. MFRC presented a webinar entitled, "Metal forming simulation using AFDEX."