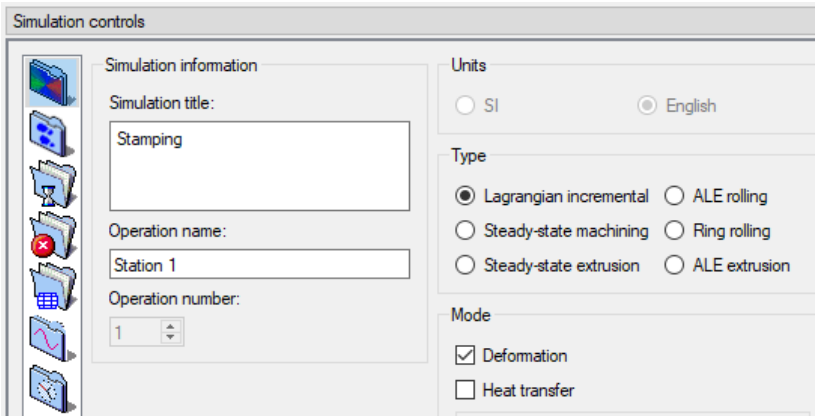
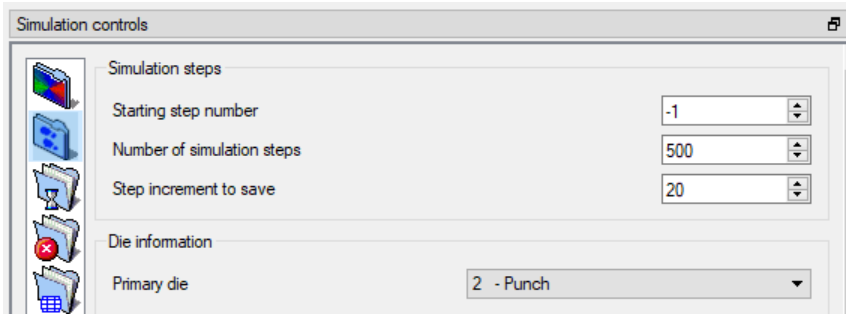
MODELING SHEET METAL FORMING USING DEFORM®

The following document provides best practices for modeling sheet metal stamping and deep drawing processes using DEFORM. The instructions are applicable to both DEFORM-2D and DEFORM-3D, except where noted. A DEFORM-3D model is used as an example.

- Simulation Controls
  - Give each operation a logical name and turn off heat transfer calculations.

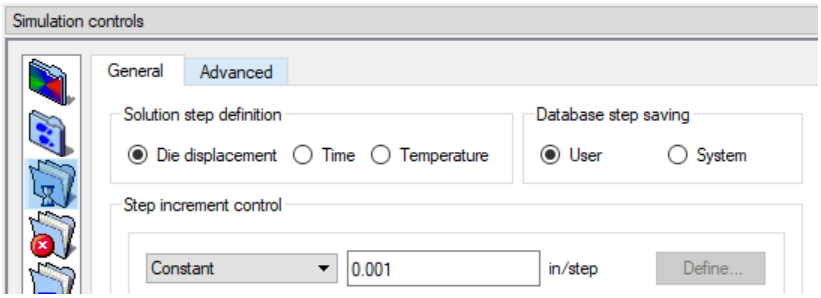


  - In forming operations, specify a large number of steps. It should be much more than needed. The stopping controls will be used to end each operation.

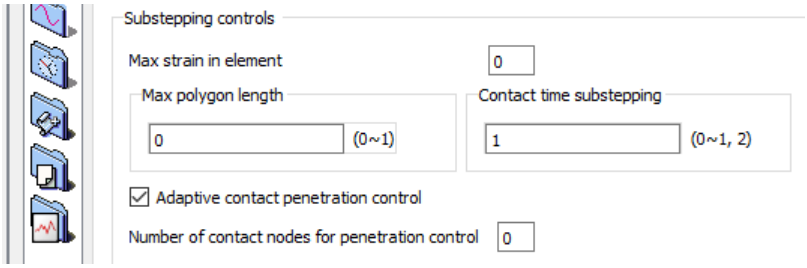


  - In sheet forming simulations, particularly elasto-plastic (E-P) models, set the die displacement step size to be 5-10 times smaller than the minimum element thickness. In this case, we will use 0.001 in/step.
  - Substepping behavior may optionally be altered by changing “Database step saving” from User to System. The “User” option may lead to long substep calculation times, since the reduced time increment that was used in the first substep is repeated until the end of the full step. The “System” option can significantly improve runtime by ending the full step after the first converged substep calculation. The downside to the “System” option is that it results in

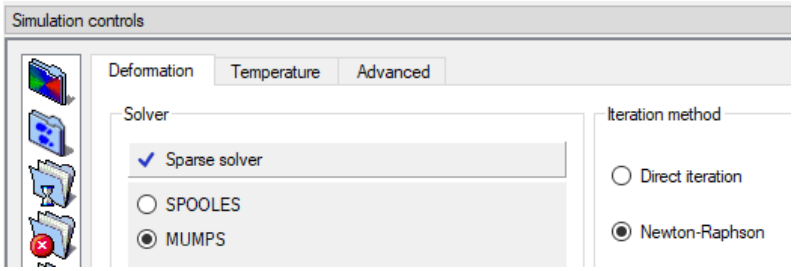
nonuniform displacement and time step increments for the steps saved to the database. See the DEFORM user manual for more information on the subject.



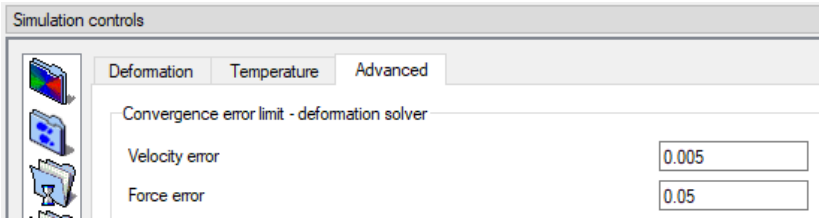
- It is recommended to deactivate polygon length substepping in 3D E-P simulations by setting the “Max polygon length” to zero. This is not applicable to 2D simulations. While this setting is beneficial in typical bulk forming simulations, its activation is not well suited for E-P sheet metal forming simulations.
- In 3D, turn on “Adaptive contact penetration control”, which activates implicit contact controls for better E-P contact handling. See the DEFORM User Manual for a description of the feature. It provides better volume control and may also result in better convergence behavior in sheet metal applications. This menu option is currently only available in 3D. The DEF_CNT.DAT file is used to activate the feature in 2D. Details on the DEF_CNT.DAT file are provided at the end of the document. The “Do not update” node coordinate updating setting, which is described later, should always be used when the “Adaptive contact penetration control” feature is active.



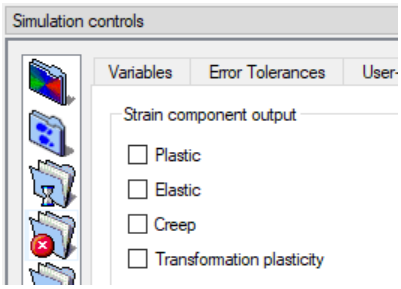
- Use the MUMPS solver and the Newton-Raphson iteration method.



- It is recommended to start with the default velocity and force error norm limits, which are 0.005 and 0.05 in 3D. A smaller force limit will produce higher accuracy, but will lead to longer runtimes and more risk of nonconvergence. The value may be temporarily increased as one approach to get through periods of convergence difficulty. Node pulling (spikes in the mesh) and other undesirable response may occur randomly during the simulation if the limit is too large.



- Plastic and elastic strain components may optionally be saved to the database by selecting them for strain component output. The selected output results will then be available in the postprocessor. The default settings only store total strain data for use in the postprocessor.

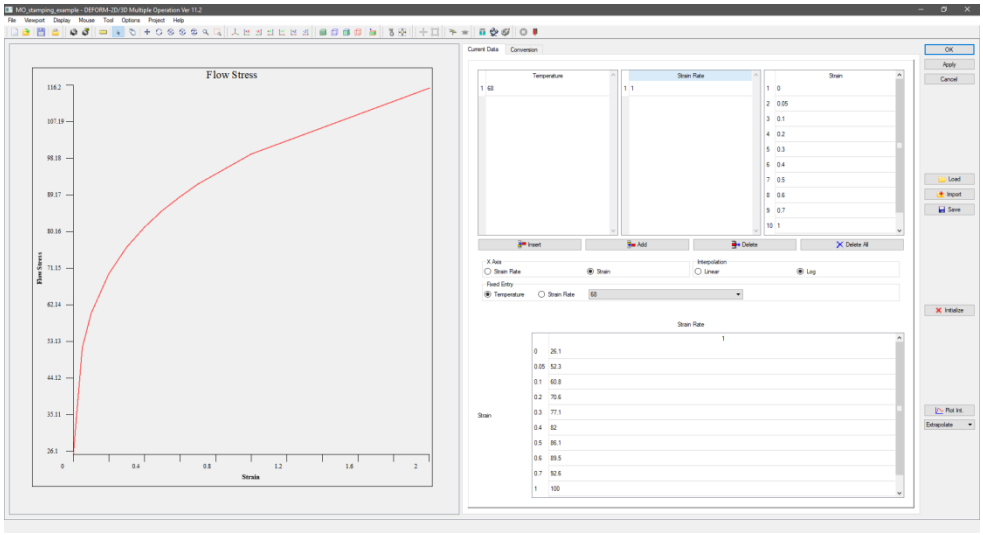


- In the same menu, it is recommended to change the state variable interpolation method to “Element + node”. This change reduces interpolation error and better maintains state variable results across many remeshes.
- Augmented Lagrangian Contact (ALC) is an optional alternative to the default (recommended) penalty contact method. It is specified through the “Simulation controls > Advanced options > Contact” menu. It might provide better convergence and runtime in certain models, though this merits more tesitng. It may be used with “Adaptive contact penetration control” and the “Do not update” node coordinate updating features.

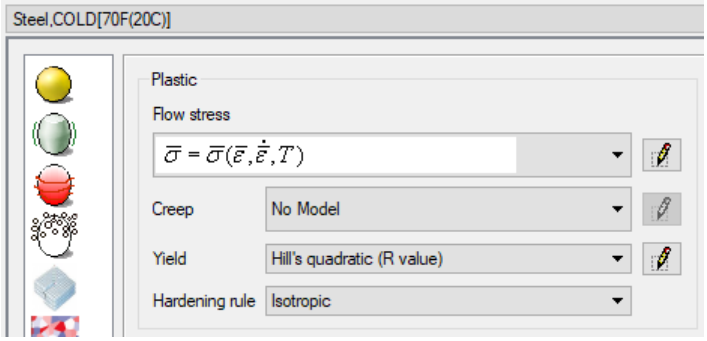
Material

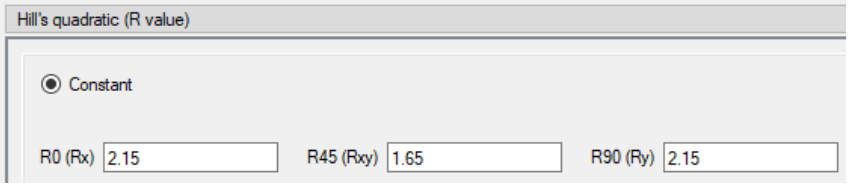
- When dealing with room temperature (cold) material, it is recommended that the user creates a plastic flow stress curve based on the yield and tensile strengths of their particular material, grade and condition. This curve can be generated using a flow stress spreadsheet that is available from DEFORM support staff.

- A flow stress curve representing a single temperature and single strain rate is ideal for E-P solutions. Curves for multiple strain rates may be used if the material's flow stress is significantly sensitive to strain rate.



- In 3D models, planar anisotropic behavior may be captured by setting the yield criteria "Hill's quadratic (R value)". The r0, r45 and r90 parameters must then be defined for the particular material. The relative difference in the values dictates the amount of earring in sheet forming operations.

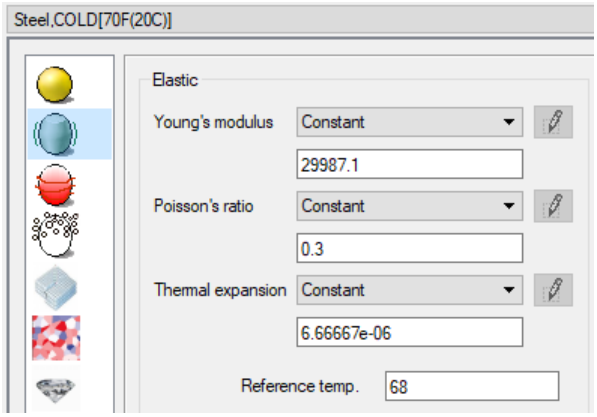




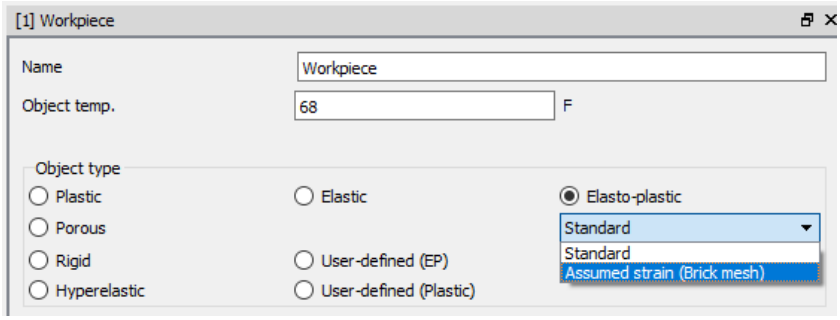
- The r0, r45 and r90 directions are aligned with the X, XY and Y directions in global space, as a default assumption. This is defined by the "material axes" in the properties for each element. If the workpiece is not oriented with the forming

direction in the Z direction, then the material axes normal must be adjusted for all elements. This can be accomplished through the element properties menu.

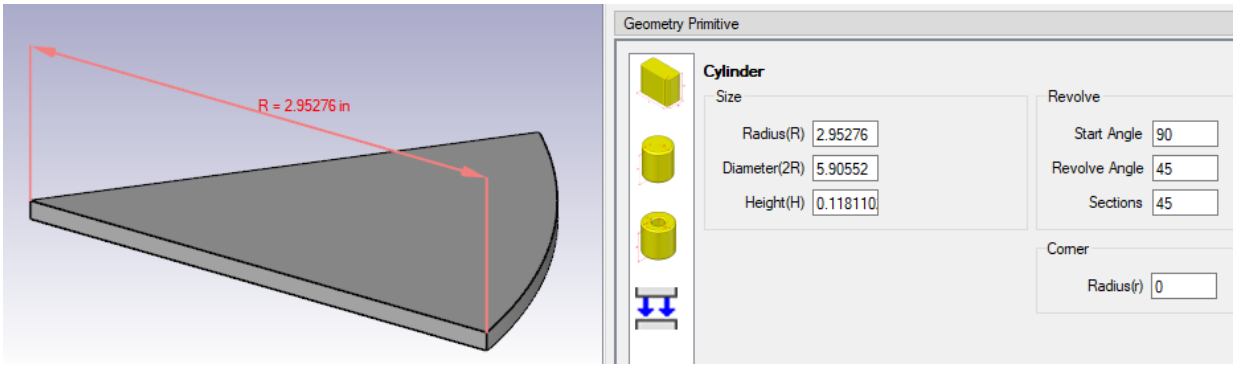
- Elastic properties must be defined if the material is used on an E-P object.



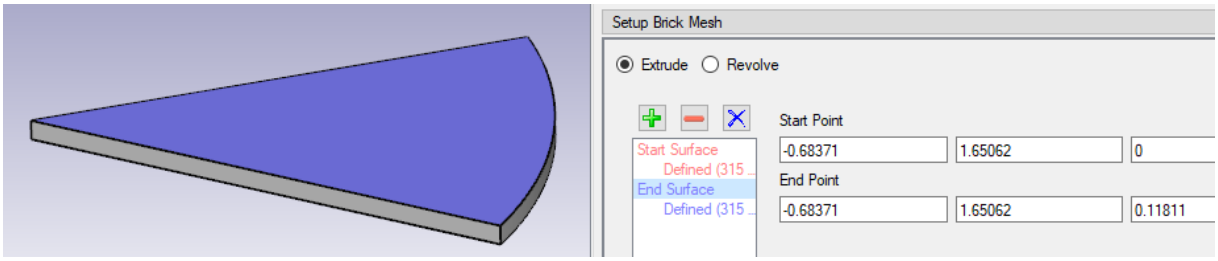
- Workpiece object
  - Model the sheet metal workpiece using the E-P material model. The plastic material model is less desirable because it does not model springback.
  - The "Assumed strain (brick mesh)" E-P element formulation should be selected if a thin sheet metal model only has 1-2 brick elements through the thickness. The setting can improve simulation accuracy for such models. It is particularly useful for forming processes that involve a lot of bending and not much stretching. The "Standard" formulation may be used if the model experiences both bending and stretching or stretching alone. The "Standard" formulation should be used if the sheet metal is modeled with 3+ layers of brick elements through the thickness.



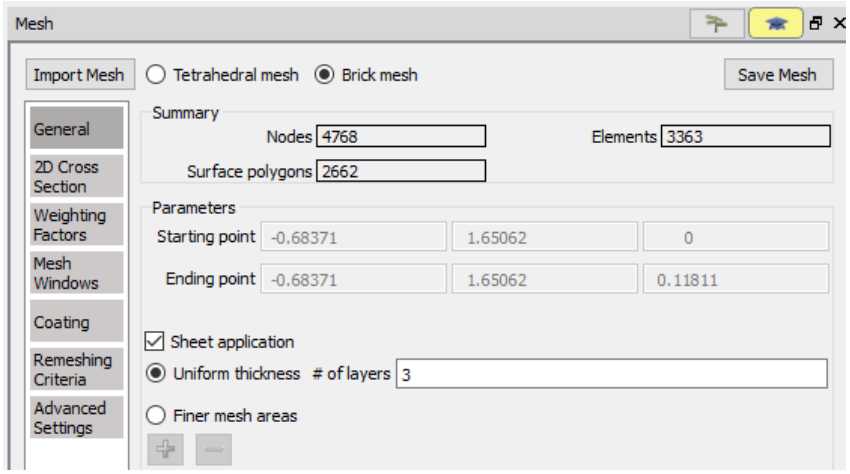
  - Workpiece and die objects should be created with the forming direction aligned in the Z direction for 3D models and the Y direction for 2D models. This avoids issues with material axis orientations when anisotropic behavior is specified.
  - Create the sheet metal blank using the appropriate geometry primitive. Symmetry may be incorporated, if desired. Editing tools are available to refine the 2D cross-section, using XY point manipulation, if needed.
  - Specify the symmetry planes on the 3D geometry, when applicable.



- For 3D geometry, use “Setup brick mesh” menu to define extrusion start and end faces. This is necessary for extruded brick mesh generation.

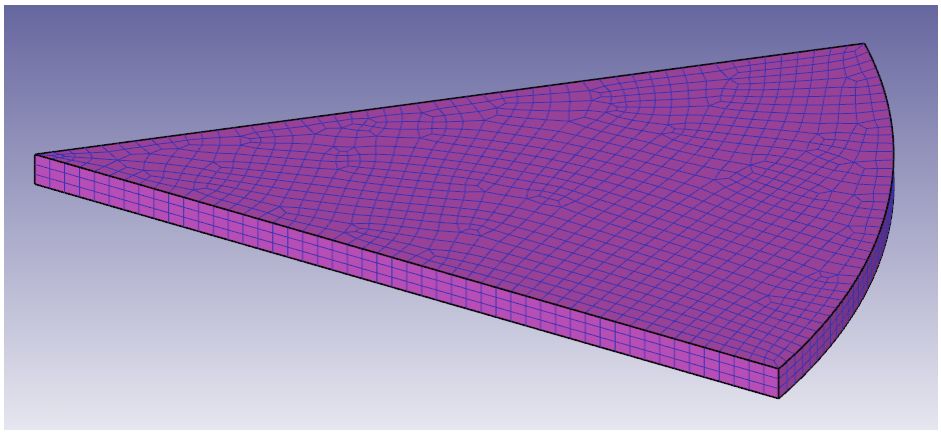


- An extruded brick mesh is preferred in 3D. Tet meshes may be used if the shape becomes too complex for brick remeshing.
- Three or more layers of elements are recommended through the thickness of the part, in order to best capture through-thickness behavior. One to two layers of elements may be used as a simplification. As described above, the “Assumed strain (Brick mesh)” setting should be selected in a 1-2 element layer model.
- Turn on the “Sheet application” option, which activates projection-based brick remeshing for sheet metal, brick meshed objects. The setting only applies to an object that was initially generated as 2D mesh and then extruded into 3D mesh.

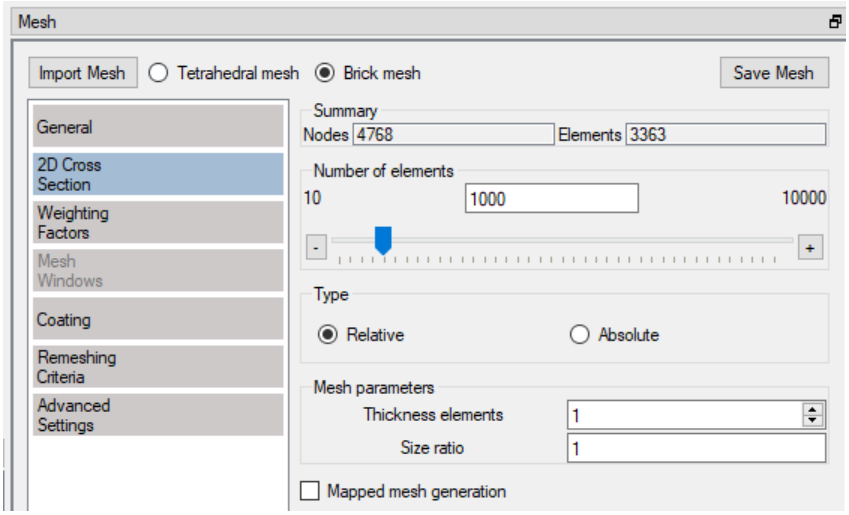


The “Sheet application” remeshing procedure requires the total nodal displacement values that have accumulated since the beginning of the simulation, when the geometry was a flat sheet shape. Do not initialize displacement when modeling a process sequence. The sheet remeshing procedure will not work properly if displacement is initialized.

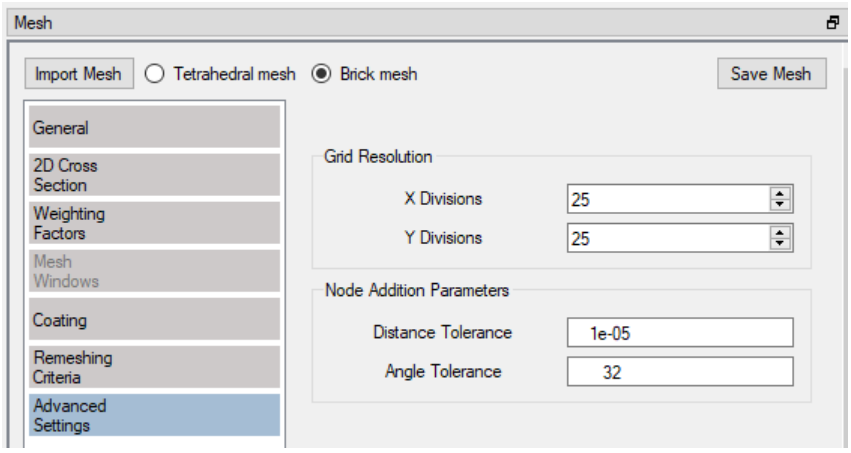
If brick remeshing fails, then the mesher will fall back to tet elements. Tet remeshing will also be performed if the initial mesh is brick but the “Sheet application” option is deactivated (default).



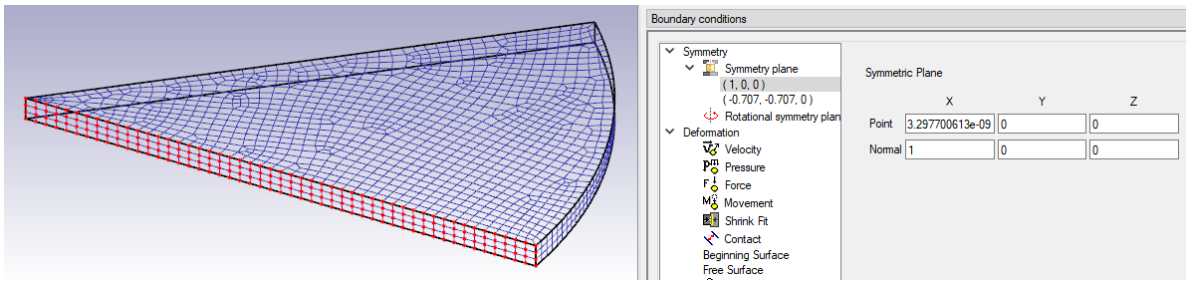
- 3D brick meshes are created by extruding a 2D mesh into 3D. The number of 2D elements therefore dictates the amount of mesh detail on the face of the sheet. The number of 2D elements should be tailored for a specific project’s accuracy, detail and simulation time requirements. The thickness elements and size ratio should both be set to 1, so a uniform mesh is produced on the 2D area.



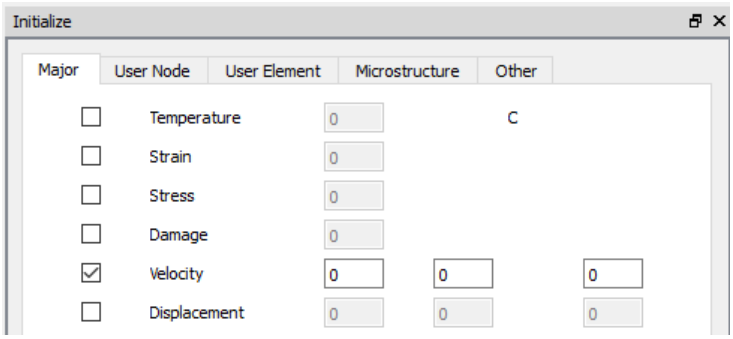
- Many sheet metal parts simulated with DEFORM are of small size. It is thus recommended to reduce the 2D node addition distance tolerance from the default value. A tolerance of 1e-5 is suitable for most sheet metal purposes.



- If 3D workpiece symmetry is being utilized, then define symmetry boundary conditions on the cut faces.

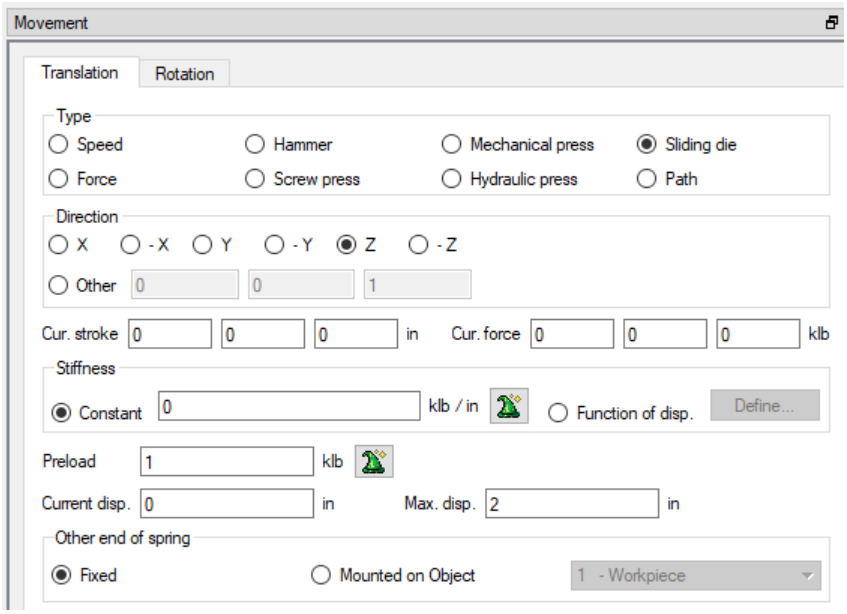


- Initialize the workpiece velocity at each operation. This improves convergence at the start of the operation.
- <u>Do not</u> initialize displacement when the “Sheet application” meshing option is utilized. The total nodal displacement values, accumulated since the geometry was as a flat sheet, are required for the sheet remesher to work properly.



- Die objects
  - “Mark” the symmetry faces on the dies to prevent nodes from possibly coming into contact with the symmetry surface. This is accomplished by defining each symmetric face as a planar symmetry surface from within each die object’s geometry menu.
  - Punch movement may be defined as normal. An optional simplification, if the material is not strain rate sensitive, is to set the punch movement to a constant speed. One trick to possibly improve convergence in problematic E-P models is to arbitrarily set the die speed to a value that produces a more nominal time step. In some models, very small time steps (maybe 1E-5 or less) might cause difficulties with convergence. In some of these situations, convergence can be improved if the time step is adjusted to be a few magnitudes larger. This adjustment is performed via manipulation of the die speed.

- Sliding die objects
  - Sliding dies are useful for modeling pressure pads and spring-controlled dies. They are advantageous for applying a force and having defined stopping and starting limits. They can also be set to travel in conjunction with another object.

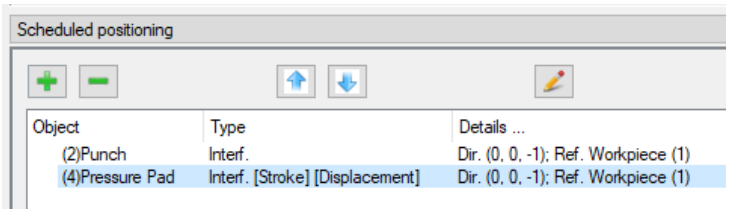


  - In the MO environment, a sliding die must be defined in its unsprung position and its current displacement must be set to zero. The user then considers whether scheduled positioning is needed. If the sliding die should start the simulation in its unsprung position, then it needs no scheduled positioning. If the sliding die should instead start the simulation compressed and in contact with the workpiece, then interference scheduled positioning should be defined. In that scenario, the checkbox to update the sliding die displacement must be selected.

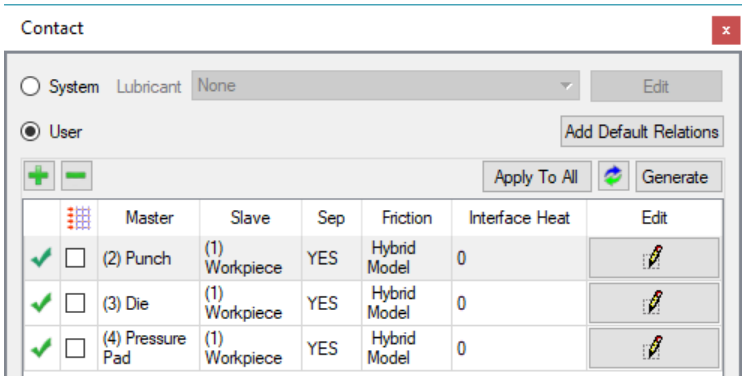
The checkbox will automatically update the spring's current displacement value based on how far the die moved (compressed) during interference positioning.

- Pressure pads of constant force may be modeled by defining a preload value and zero spring stiffness.
- Fractional preload and stiffness values must be input for 3D models that use symmetry. For example, when one-eighth of a workpiece is modeled then the preload and stiffness inputs would be one-eighth the actual values.
- The maximum displacement setting defines a limit for die travel in the compression direction. This must be defined in MO simulations where the sliding die's current displacement is set to automatically update during scheduled interference positioning.
- The sliding die will never move to a position where its "displacement" value goes below zero. The spring will be fully uncompressed when the displacement returns to zero. This provides a limit to the die travel in the extension direction.
- Multiple sliding dies may be used as long as they are set up as described above.
- It is recommended to change the sliding die "Number of iterations" limit from the 20 to 200. This effectively deactivates spring damping on the sliding die, which may improve convergence. The NOSPRDMP.DAT file (described later) may alternatively be used to fully deactivate spring damping.

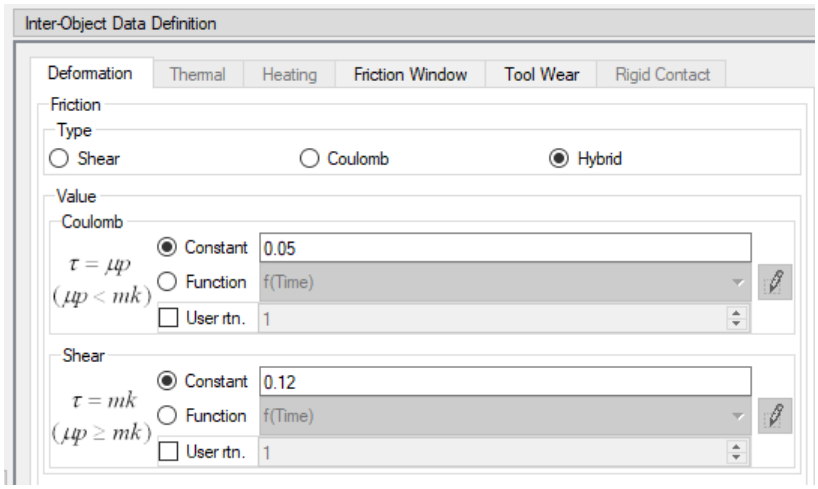
- Positioning
  - Model the tools in the closed position. Use scheduled positioning to get them to the initial state. Sliding dies are the exception, as addressed above. The following image shows a scheduled positioning for a forming operation. In this case, the sliding die (pressure pad) is being brought into contact with the workpiece. If the sliding die will instead start the process in its unsprung position, without workpiece contact, then the pressure pad interference positioning shown below should not be defined.



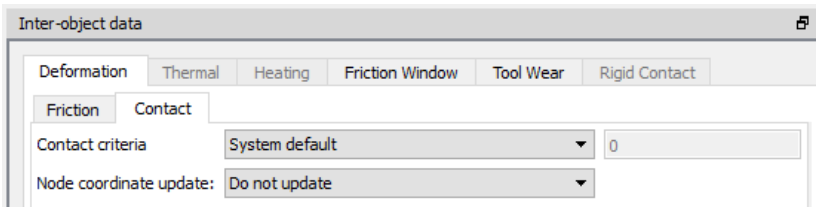
- Contact
  - In forming operations, add contact pairs between each of the tooling objects (master) and the workpiece (slave) object.



- Apply hybrid friction for all object pairs. Recommended settings are coulomb friction = 0.05 and shear friction = 0.12.

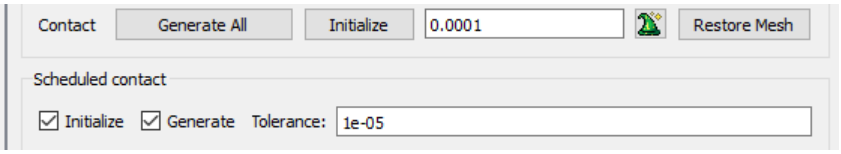


- Within the “Contact” options, select “Do not update” for “Node coordinate update”. It deactivates contact node coordinate updating. This prevents slave nodes from being pushed to a master surface when contact is determined. This should be used with “Adaptive contact penetration control” in 3D. It may be used alone in 2D. This setting replaces the “DEF_EPCON.DAT” file.

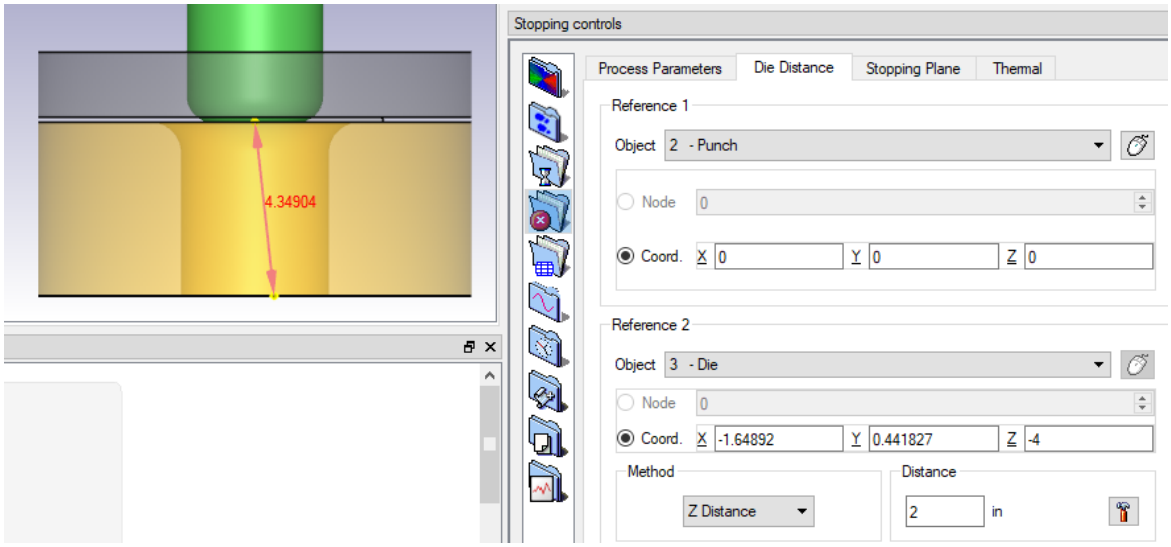


- Define a scheduled positioning contact tolerance that compliments the geometry. Some thin sheet metal parts require a very small tolerance to avoid situations where the automatic tolerance might cause node pulling. In other cases, a larger

tolerance may be needed to guarantee that at least some initial contact is generated between the workpiece and various dies. In the lab example, a value of 1e-5 is utilized to avoid node pulling.

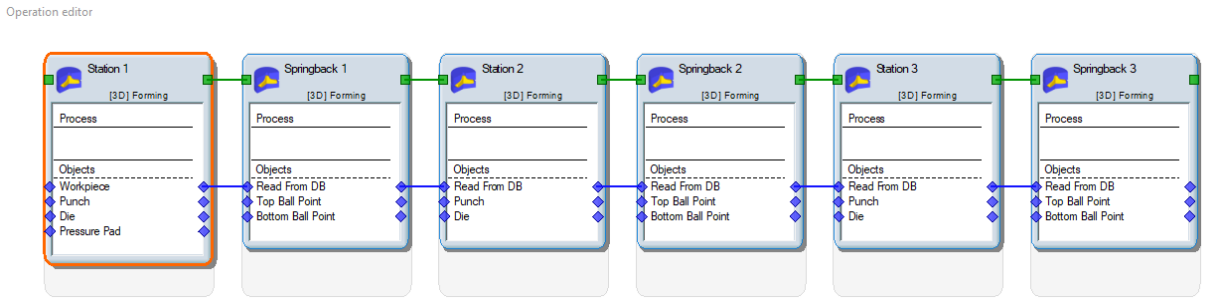


- Stopping controls
  - At least one stopping control must be defined. Multiple stopping controls may be desirable in some situations.
  - Avoid using the stroke stopping control in MO projects, since it often changes when geometry and process conditions change. A die distance or load based stopping control is more flexible and thus recommended.

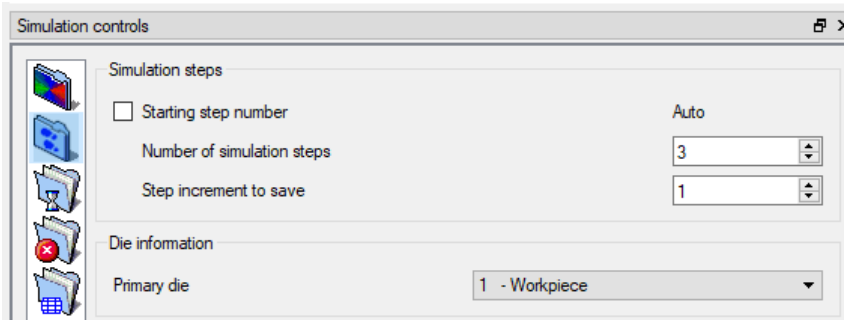


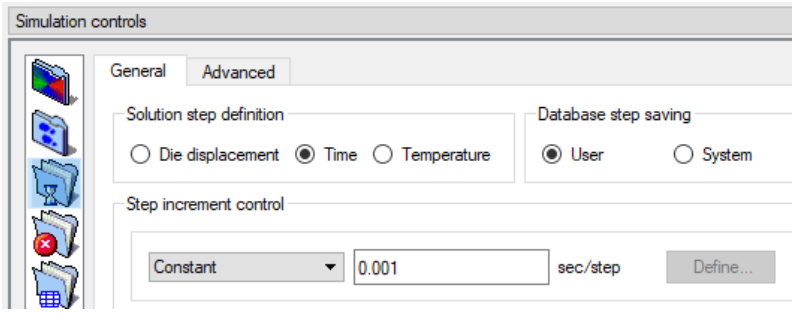
  - The die distance stopping control is ideal for sheet metal forming operations. Distance is measured between reference points on the target objects. Object reference points may be selected by clicking the mouse button, which allows interactive picking. Reference points can also be set, during object definition, through the object's "Properties > Reference" menu. The measurement "Method" should be set to "Z Distance" since the forming direction is oriented in the Z direction. The simulation will stop when the specified closure distance is reached, as measured in the defined direction. This stopping control acts from a large distance down to a smaller (target) distance, not the other way around. Stopping distances may thus need to be defined from a point low on the die.

- Springback operations
    - Include a springback operation after each sheet metal forming operation.

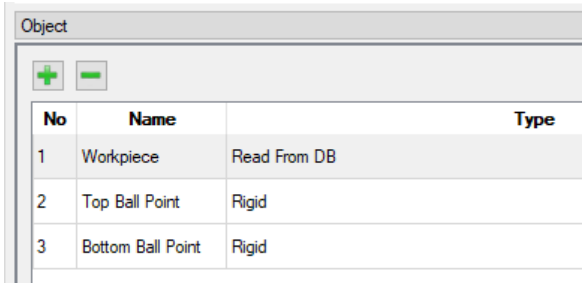


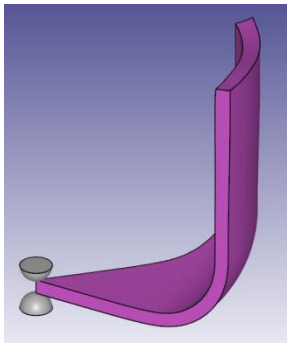
    - Springback should run for 1-3 steps with a time step size of 0.001 seconds.



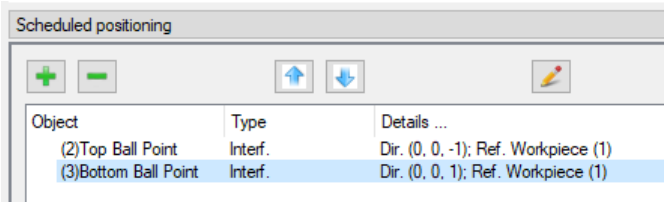


    - Two ball point tools may be used to provide stable, yet minimal, constraint in springback operations. Please contact your DEFORM support representative for sample ball point geometries. These may be scaled to appropriate size.

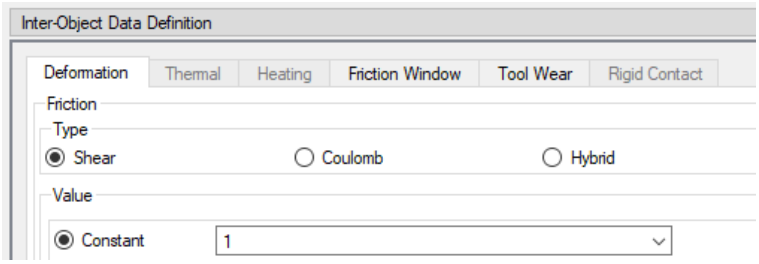




- Use scheduled positioning to have the ball points lightly touch the workpiece from above and below.



- Use shear friction = 1 between all object pairs in springback operations. The workpiece may also be set as non-separable.



- DAT files (Add to the working directory, if desired, as described below.)
  - Optional for E-P and/or sheet metal simulations
    - DEF_CNT.DAT (2D only)
      Changes the contact behavior to prevent nodes from pushing to a die surface during updating. This provides better volume control and may also result in better convergence behavior in sheet metal applications. The file is not critical in 2D, since contact substepping limits penetration of the workpiece into a die. The text file should contain one line followed by a carriage return. The first line should read “1”. The file had previously been used in 3D, but it has since been replaced by the “Adaptive contact penetration control” option in the Simulation Controls menu.

        - DEF_EP.DAT
          Enforces force and velocity error norm convergence checking on E-P models. Both error norms used by default, but the velocity error norm may be ignored if convergence difficulty is encountered. Use of the velocity error norm improves accuracy and often prevents mesh distortion or failure due to poor velocity profiles. Try adding the file if node pulling or mesh distortion issues occur.
        - DEF_INCDISP.DAT
          Activates incremental displacement solution control. The setting is not recommended as a default. Though, it may be useful when troubleshooting convergence issues. It may particularly improve convergence during the middle of a deep drawing simulation, when material transitions from pulling inward to moving in the forming direction.
        - MDIR.DAT
          Deactivates the Direct iteration method. Direct iteration may produce accuracy issues in elastic-plastic models if the solver switches to it every step. It is okay to have occasional switching to direct iteration, as long as the majority of steps are solved using the Newton-Raphson iteration method.
    - Optional for simulations involving sliding (spring) dies
        - NOSPRDMP.DAT
          Deactivates spring damping when sliding dies are present. Excessive spring damping may lead to convergence issues.
        - SPR_DIRCHK.DAT
          Alters spring direction checking behavior. It prevents scenarios where the solution may not converge when a workpiece only contacts the side of a sliding die. In other words, when the contact normal is nearly perpendicular to the direction of spring travel.

<u>Revision History</u>

- Rev. 8 – 5/3/2024
    - Renamed document from stamping to sheet metal forming focus.
    - Updated to follow V13 guidelines and best practices.
    - Added guidelines on substepping database saving controls.
    - Revised the “polygon length substepping” description.
    - Updated the “adaptive contact penetration control” and DEF_CNT guidelines.
    - Revised the “force error norm” guidelines.
    - Revised “strain component output” information.
    - Added “Augmented Lagrangian Contact (ALC)” guidance.
    - Revised the E-P element formulation descriptions and recommendations.
    - Updated recommendations for the number of element layers in a brick mesh.
    - Revised the “sheet application” meshing information.
    - Changed the recommendation regarding whether to initialize displacement.

- Added comment on die object symmetry surfaces and punch speed.
- Added sliding die iteration limit instructions.
- Revised contact tolerance guidance.
- Moved MDIR.DAT and DEF_INCDISP.DAT files from recommended to optional.
- Updated the DEF_CNT.DAT and DEF_INCDISP.DAT file descriptions.
- Other minor changes.

- Rev. 7 – 2/22/2023
  - Added “Adaptive contact penetration control” setting.
  - Added “Assumed strain (brick mesh)” information.
  - Added the “Sheet application” setting for brick remeshing.
  - Updated “Contact” settings information including that regarding contact pairs, node coordinate updating and contact tolerances.
  - Revised the “DEF_CNT.DAT” information.
  - Removed the “DEF_EPCON.DAT” information.
  - Revised the “DEF_EP.DAT” information.

- Rev. 6 – 10/31/2018
  - Added the revision history.
  - Added the “Max polygon length” setting.
  - Revised the “DAT File” section to better describe which DAT files are recommended versus optional. Some DAT file descriptions were also updated.