Introducing boundary conditions for turbomachinery modeling

SIEMENS

Introducing boundary conditions for turbomachinery modeling

This article describes how to define boundary conditions to simulate the complex physics in turbomachines.

Content

- **Boundary conditions summary**
- **Modeling the 1D duct network**
  - **Two modeling approaches**
  - **Convection coupling of duct’s nodes and elements to the rotating wall**
  - **Convection coupling of ducts inside a 3D solid body**
  - **Constraining duct temperatures on nodes or elements**
- **Modeling thermal contacts**
- **Specific convective boundary conditions for turbomachinery**
  - **Thermal streams**
    - **Modeling total temperature effects on wall due to rotating flow**
    - **Connecting thermal streams**
  - **Thermal voids**
  - **Thermal convecting zones**
- **Protective layers**
- **Radiation**
- **Structural boundary conditions**

Boundary conditions summary

Thermal boundary conditions

Simulation objects:

- Duct Flow Boundary Conditions
- Junction
- Thermal Couplings
- Thermal Deactivation Set
- Immersed Ducts
- Radiation

Constraints:

- Temperature

Loads:

- Thermal Stream
- Thermal Void
- Thermal Convecting Zone
- Heat Generation (Thermal Loads)

Shared boundary conditions

Simulation objects:

- Edge-to-Edge Glue/Contact
- Surface-to-Surface Glue/Contact
- Cyclic Symmetry
- Protective Layers

Loads:

- Rotation
- Gravity

Structural boundary conditions

Simulation objects:

- Structural Deactivation Set

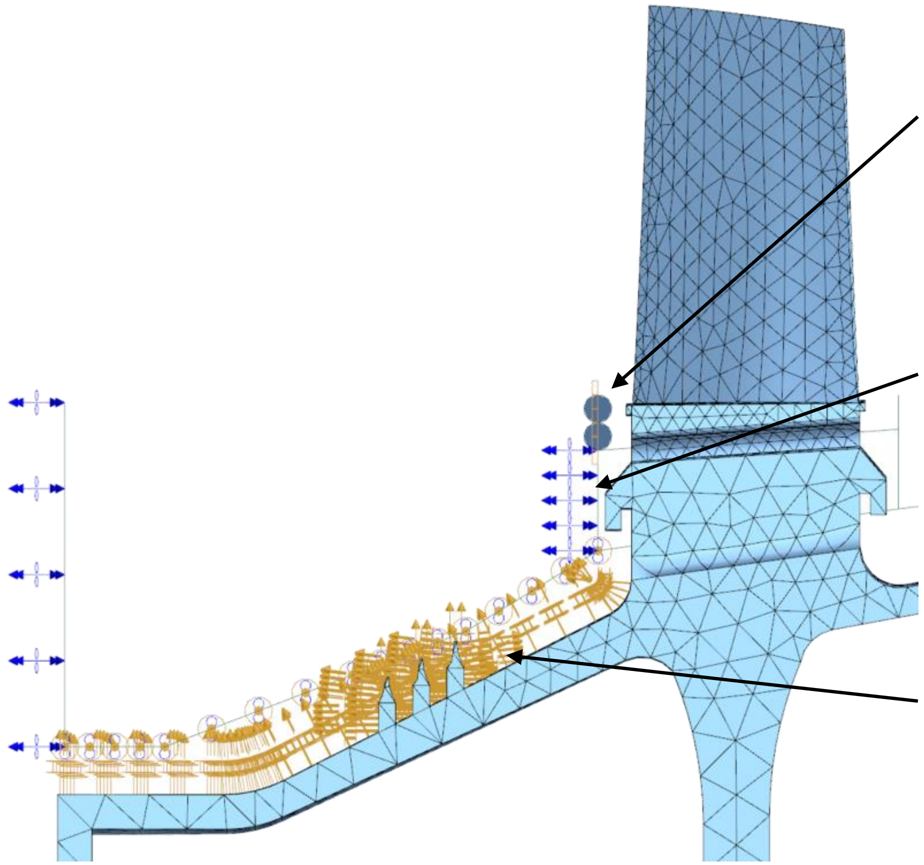
Constraints:

- User Defined Constraint
- Manual Coupling

Loads:

- Bolt Pre-Load
- Force

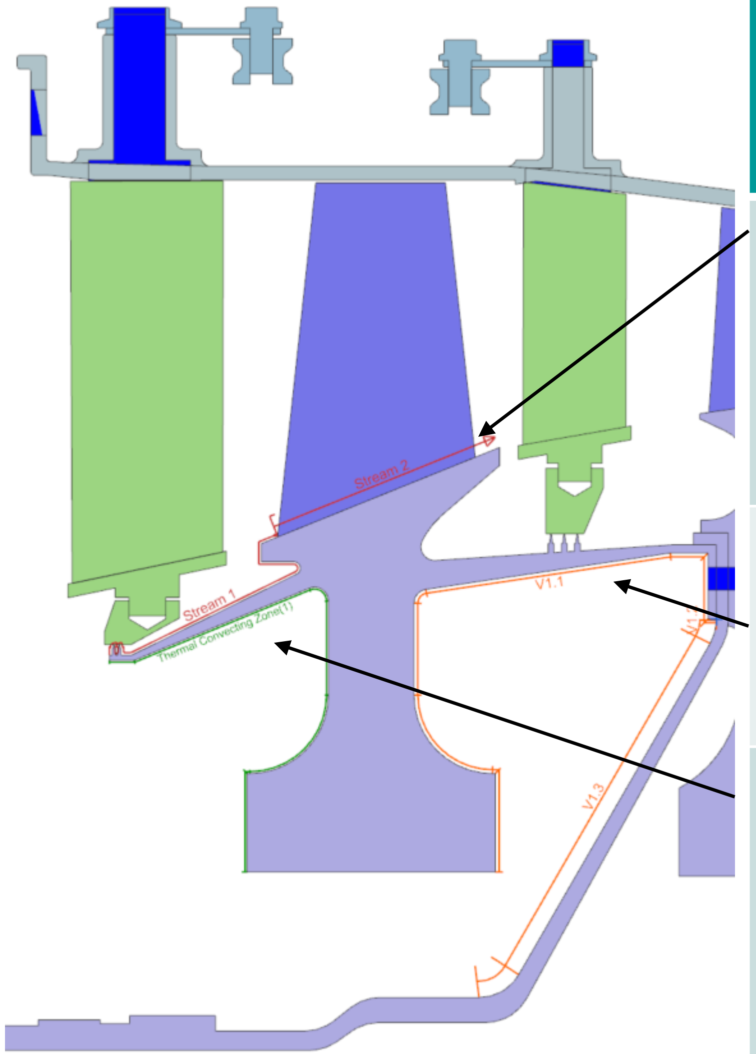
1D duct network boundary conditions for convection



Boundary Conditions	Type	Description	Quantity
Temperature Constraint	Temperature	Defines “source temperatures” for 1D fluid network.	Temperature
Duct Flow Boundary Condition	Duct Fan/Pump	Defines mass flow for fluid elements such as ducts.	Mass Flow
	Duct Label	Reference ID for other duct flows or streams.	N/A
	Duct Total Pressure Duct Static Pressure	Applies pressure to surfaces attached to duct.	Pressure
Thermal Coupling-Convection	Convection Coupling	Attaches convective surface to fluid network such as 1D ducts.	Heat Transfer Coefficient
Thermal Loads	Heat Generation	Defines heat generation (for example windage) to 1D fluid elements or nodes.	Heat Generation

SIEMENS

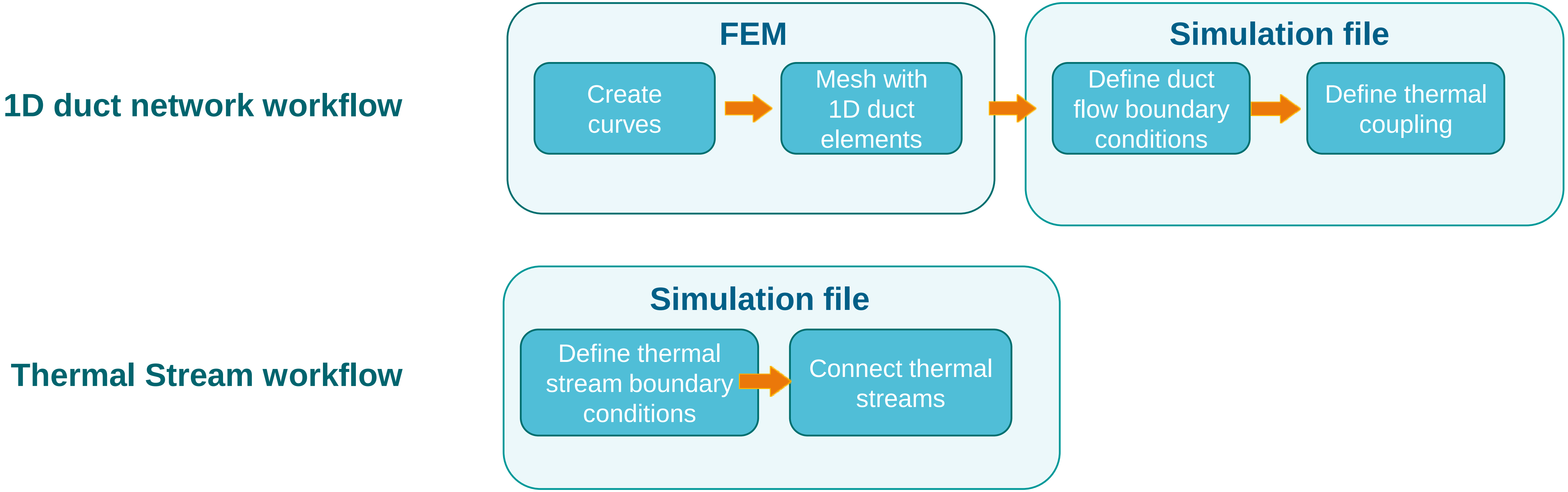
Specific convective boundary conditions for turbomachinery



Boundary conditions	Description	Quantity
Thermal Stream	Consolidation of mass flow + convective coupling + inlet temperature + heat load (Q) + pressure.	Mass Flow, Fluid Temperature, HTC, Q, Pressure
Thermal Void	“Dead” cavity with HTC and no fluid connection.	Heat Transfer Coefficient, Pressure
Thermal Convecting Zone	Convective surfaces with fixed metal temperatures such as 1-way heat transfer.	Temperatures, Heat Transfer Coefficient, Pressure

Comparing thermal streams and ducts

The purpose of ducts and streams are the same, to model convection between a fluid duct and a surface. The duct approach is more complicated since the ducts are generated by the user, while for streams the solver generates the ducts automatically. Both approaches have advantages and disadvantages. The thermal streams approach has an easier workflow and includes fewer boundary conditions, while in the 1D duct approach, the ducts can be easily compared to results from 1D flow analysis tools.



Modeling the 1D duct network

There are two general approaches to model the 1D duct network:

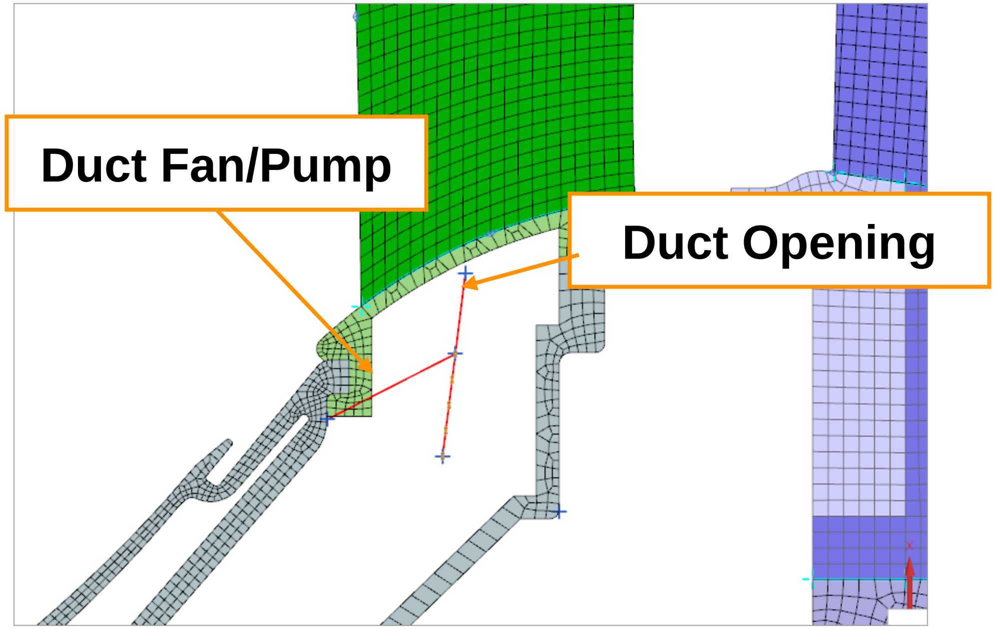
1) Applying mass flows based on external 1D results.
2) Applying pressures at openings and allowing the thermal solver to calculate flow.

Use the **Duct Flow Boundary Condition** simulation object to model 1D duct network.

Applying mass flows based on external 1D results

Use the following boundary conditions:

- **Temperature** constraint to define temperature on the node.
- **Duct Total Pressure** or **Duct Static Pressure** to define pressure on the nodes.
- **Duct Opening** to define pressure and temperature at the inlet or outlet of a duct network.
- **Duct Fan/Pump** to define mass flow in a duct network.

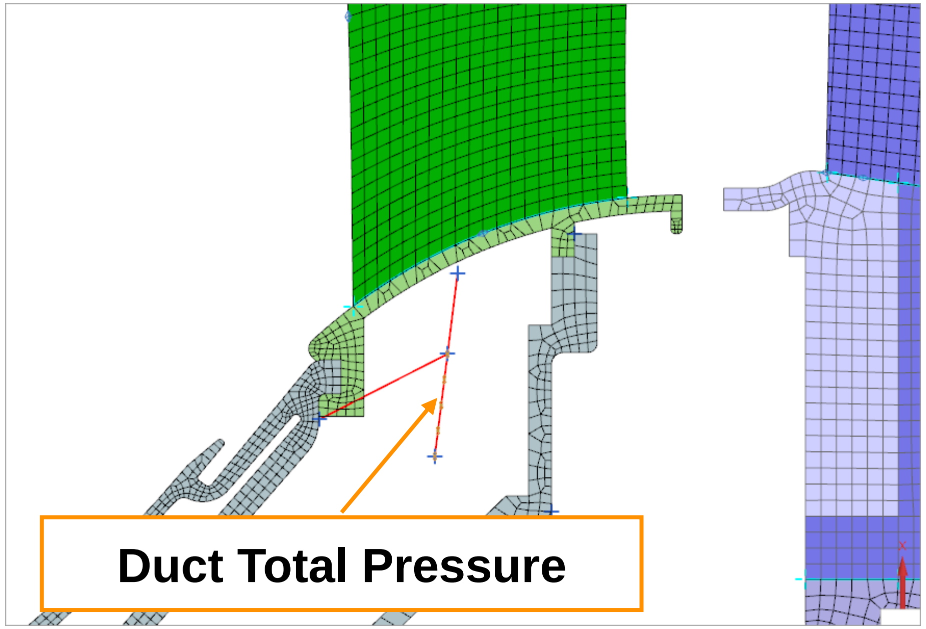


It is recommended to use the **Duct with Mass Flow** or **Duct with Mass Flow Axisymmetric** physical property when meshing the ducts. If you use the **Duct** physical property, the model will be over constrained, which will lead to unexpected results.

Applying pressures at openings

Use the **Duct Total Pressure** to apply the total gauge fluid pressure, on a point, mesh point, node, polygon edge, or curve within the duct network. The total pressure is the sum of the static pressure and the dynamic pressure.

- Applies pressure to all surfaces attached to selected ducts via Thermal Convection Couplings.



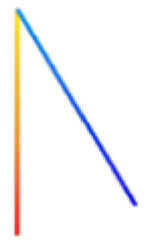
It is recommended to select curves or polygon edges when you specify fluid pressure in a duct network as it allows you to retain the boundary condition selection when you re-mesh the model.

Overriding the duct properties

Use the **Duct Flow Properties** type of the **Duct Flow Boundary Condition** simulation object to define and override the default roughness properties of the inside surface of the duct.

You can:

- Apply the roughness properties to an individual duct element, or to curves or edges meshed with duct elements.
- Model a bend or a sudden contraction or expansion using:
  - **Head Loss Override** adds pressure losses to the selected duct flow.
  - **Head Loss Multiplier** adds the constant or velocity-varying flow resistance to the selected duct flow.

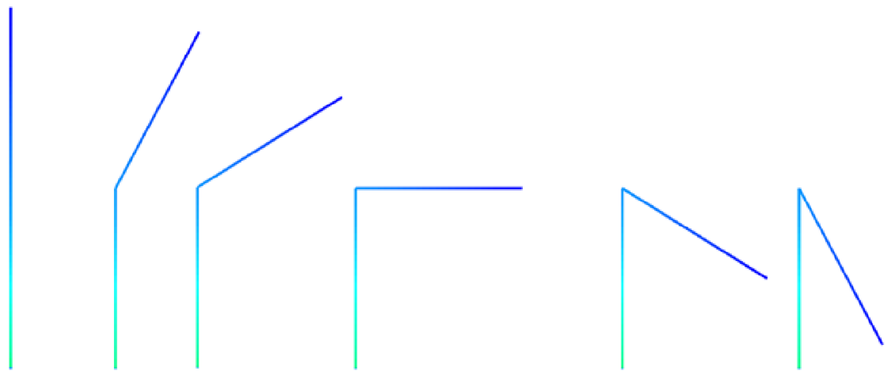


Head loss due to duct bend

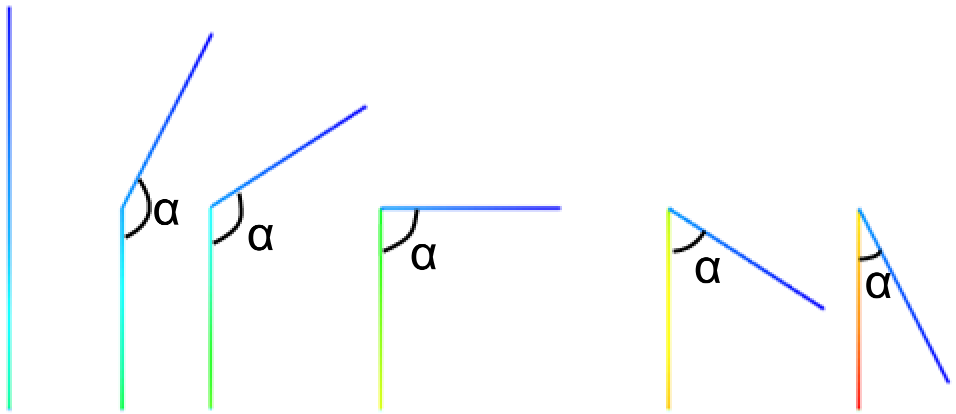
SIEMENS

Including head loss correlations for duct bends and branches

You can include pressure drop to the duct network due to curvature, bends, and junctions for all duct networks in the model when you select this option in the **Thermal Solution Parameters** dialog box. The solver computes an additional head loss and applies the loss to the duct based on angle between ducts, α.



Include Head Loss Correlations for Bends and Branches

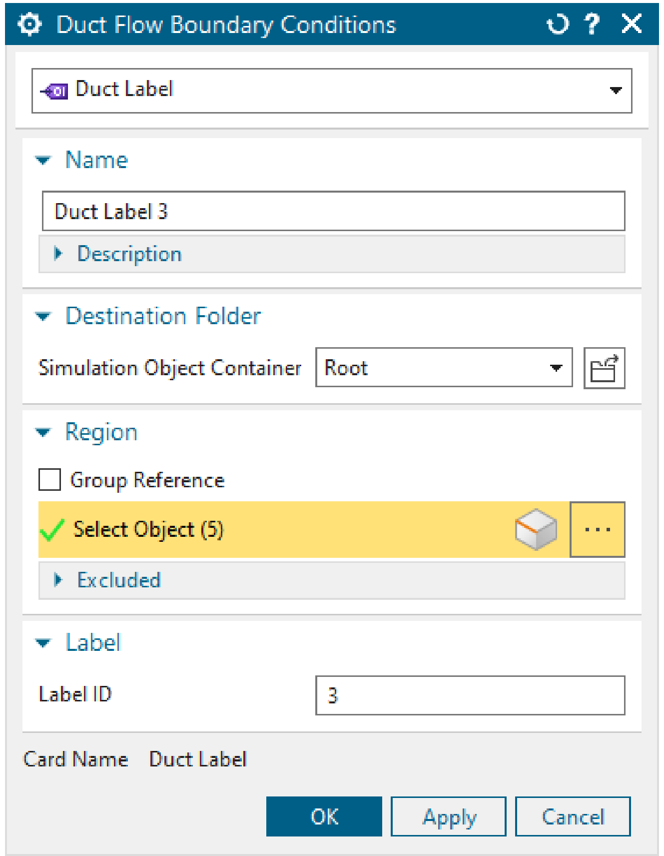


Include Head Loss Correlations for Bends and Branches

Assigning a boundary condition ID to the ducts

Use the **Duct Label** type of the **Duct Flow Boundary Condition** simulation object to assign a boundary condition ID to 1D Duct curves/elements for referencing it in the other boundary conditions using the following thermal-flow functions:

- **DMO(i)** returns the duct mass flow.
- **DPO(i)** returns the duct outlet pressure.
- **DTO(i)** returns the duct outlet temperature.



Where:

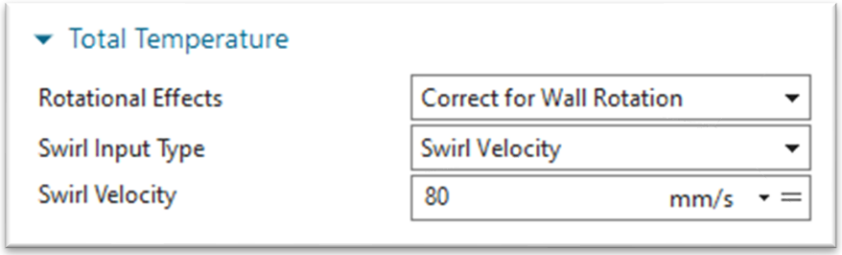
- “i” is the duct boundary condition ID.

SIEMENS

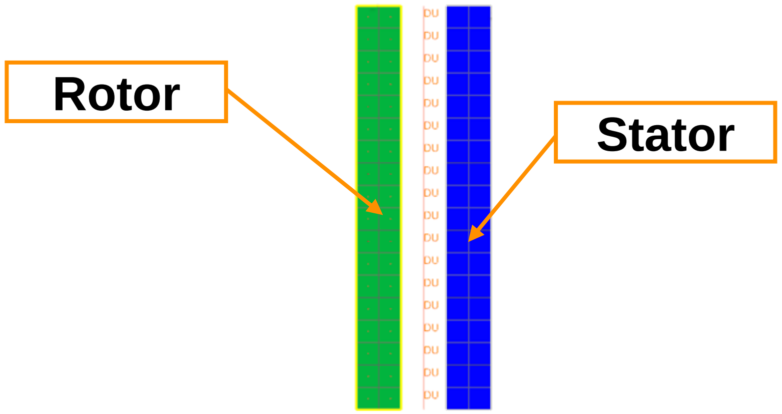
Convection coupling of duct’s nodes and elements to the rotating wall

Use the **Convection Coupling** type of the **Thermal Coupling - Convection** simulation object to model convective heat transfer between the duct’s elements and rotating wall.

You account for the total temperature effect in heat transfer between the rotating fluid and the parts using the **Total Temperature** group.



Use the **Duct Node Convection Coupling** type of the **Thermal Coupling - Convection** simulation object to model convective heat transfer between the duct’s nodes and rotating wall.

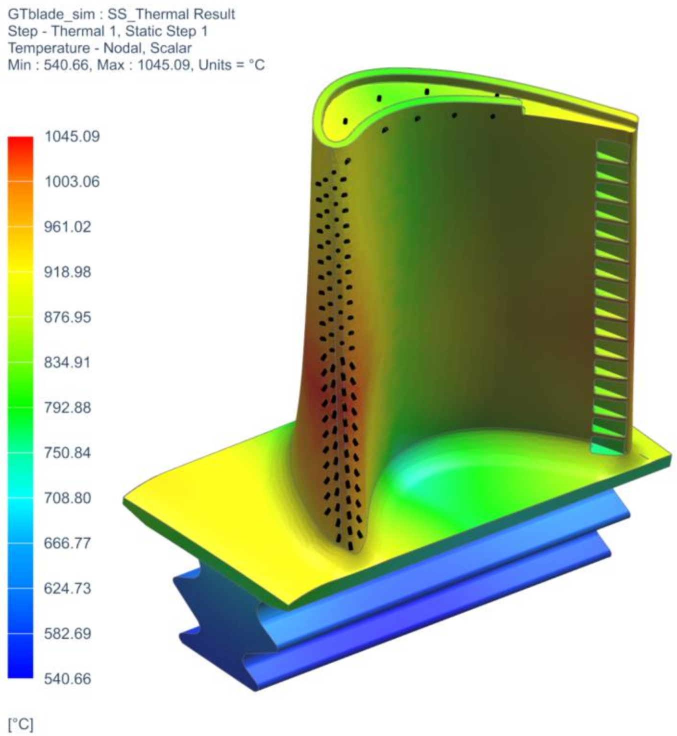


Convection coupling of ducts inside a 3D solid body

Use the **Immersed Ducts** simulation object to model the convection between the created 1D duct network, which represents cooling hole passages, and surrounding 3D solid elements without explicitly modeling the holes in the 3D geometry such as blades or vanes. Each element of a 1D duct automatically couples to a 3D solid element.

Before applying **Immersed Ducts**, you need:

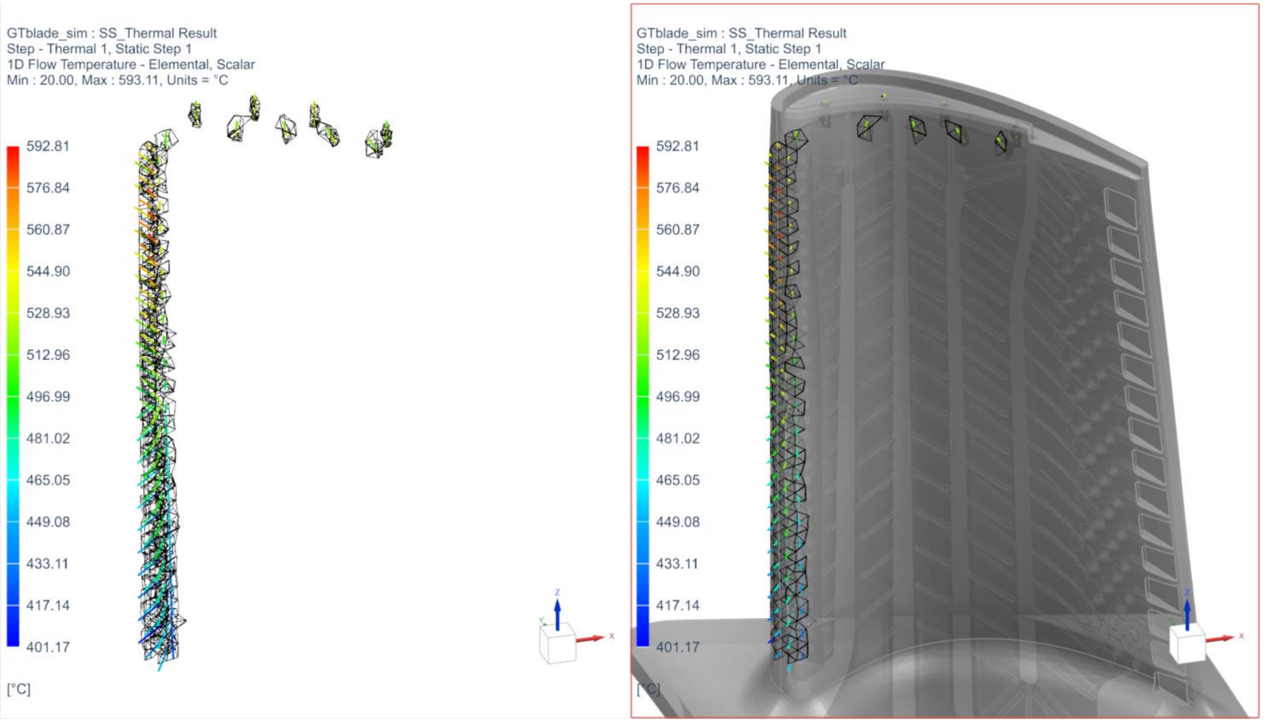
- Mesh 3D solid body.
- Crete 1D fluid elements inside the body.
- Define the **Duct** type of 1D duct mesh collector.



When you use protective layers in the model with immersed ducts, the length of 1D ducts must include the additional length equal to the protective layer thickness for correct coupling.

Visualizing immersed ducts connection results

You can visualize immersed ducts connections with the 3D solid body when you request 1D flow results. This helps you graphically identify if the immersed ducts and solid elements are correctly thermally connected when performing a 1D flow thermal analysis.



Constraining duct temperatures on nodes or elements

You can use the **Temperature** constraint to constrain the duct temperature either on nodes or elements. Constraining duct nodes' temperatures does not constrain the 1D duct elements, and vice versa. This impacts the temperature results when modeling convection coupling on constrained ducts with different simulation objects.

- **Duct Node Convection Coupling** → **Temperature** constraint on nodes
- **Convection Coupling** → **Temperature** constraint on elements
- **Immersed Ducts** → **Temperature** constraint on nodes
- **Immersed Ducts** → **Temperature** constraint on elements

Modeling thermal contacts

You can model heat paths between:

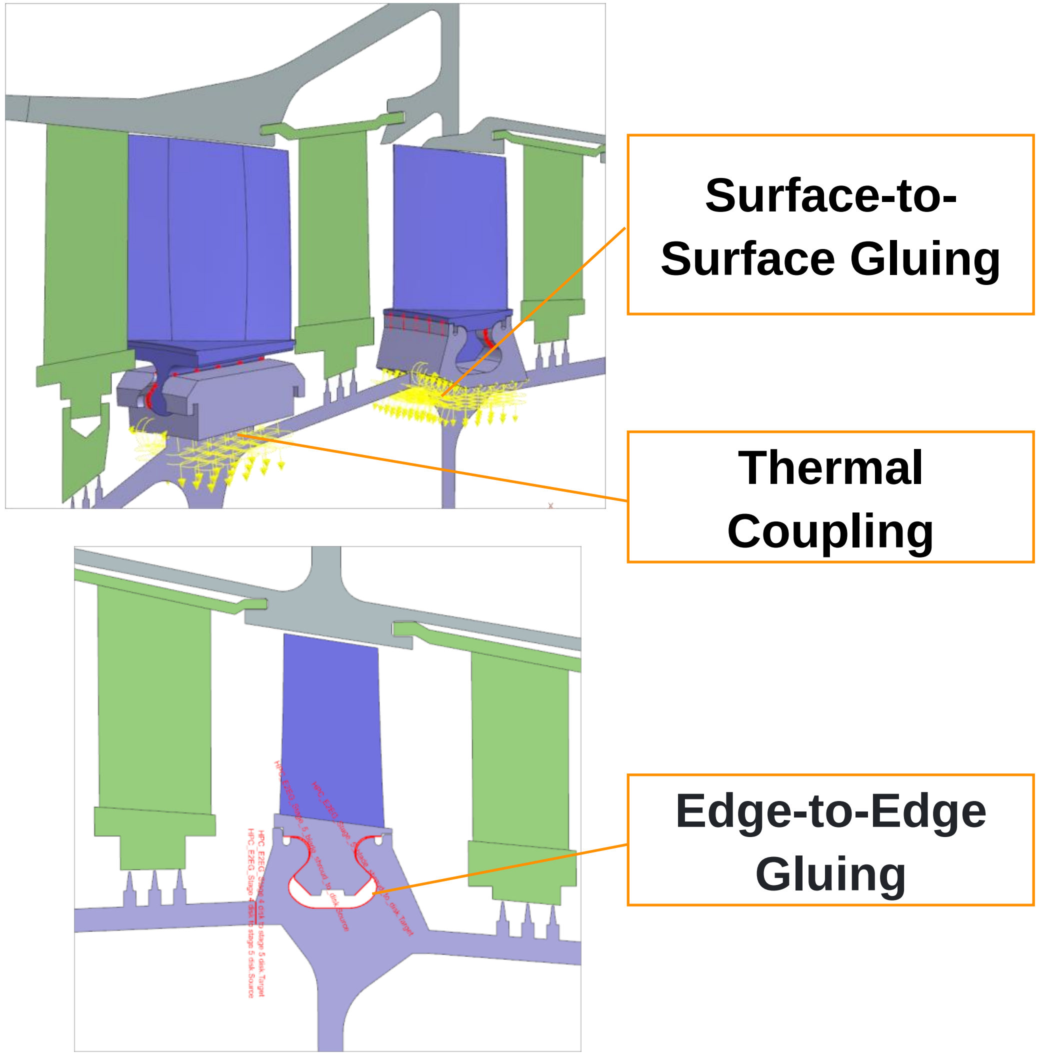
- 2D and 3D regions without having them connected through meshes using the **Thermal Coupling** simulation object.
  - Select the face of the 3D meshed body as a primary region
- 3D and 3D regions using the **Surface-to-Surface Gluing, Contact, Contact/Gluing** simulation objects.
- 2D and 2D regions meshed with 2D axisymmetric, plane stress, or plane strain elements use the **Edge-to-Edge Gluing, Contact, Contact/Gluing** simulation objects.

HTC value can vary with gap and can be defined as a single value, formula, or table.

▾ Thermal Properties

☑ Activate Thermal Coupling

Type	Thermo-Mechanical Contact
Coefficient with Gap	W/m^2-dK
Coefficient with Contact	W/m^2-dK

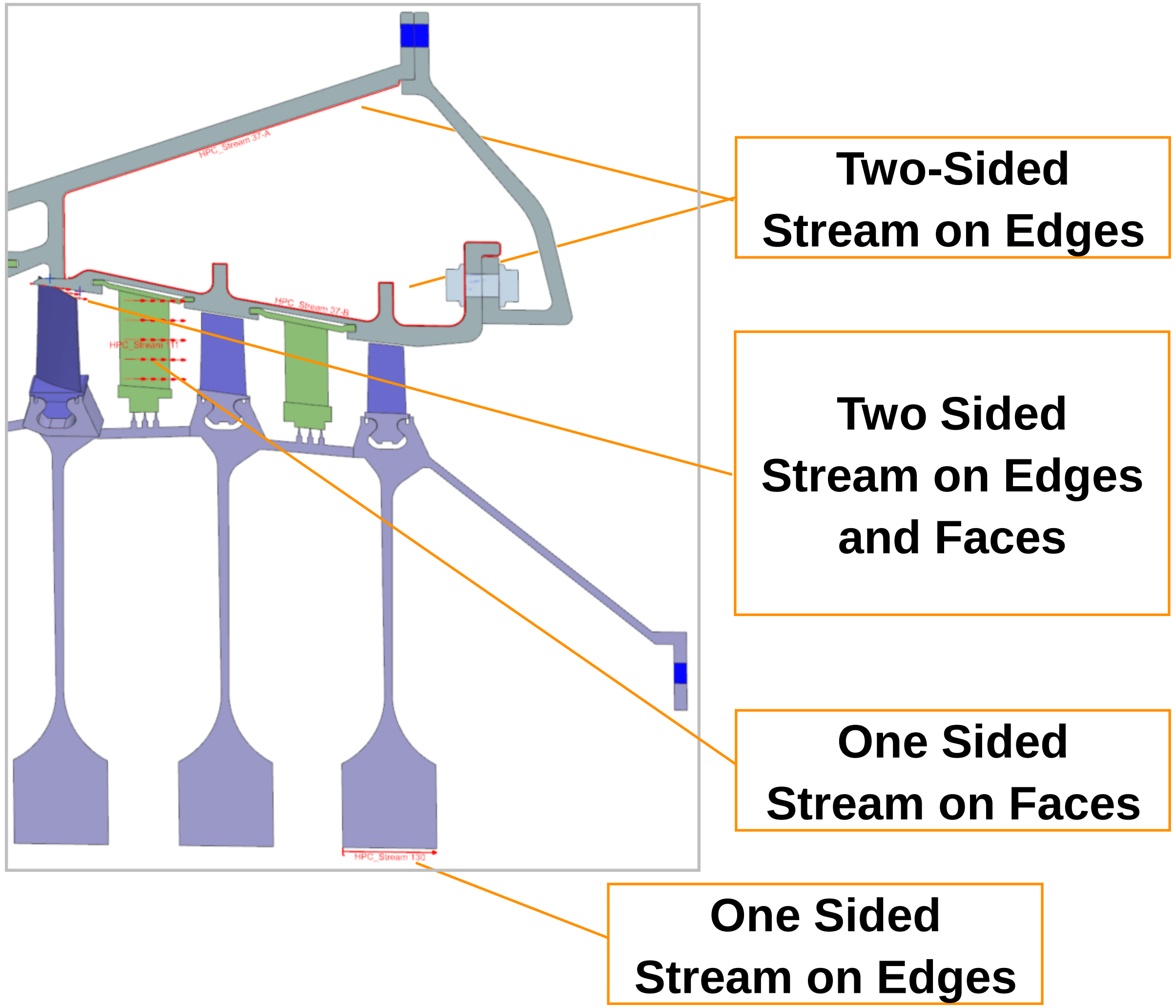


Using thermal streams

Use the **Thermal Stream** load to define convection due to fluid flow over surfaces, or over axisymmetric edges.

The thermal solver internally creates 1D duct with mass flow elements on the selected regions and connects them to the nearest thermal solid elements through a convection thermal coupling.

- No fluid network creation required
- Models a stream of air with inlet conditions and a known flow direction

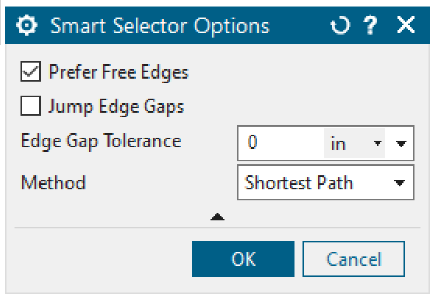


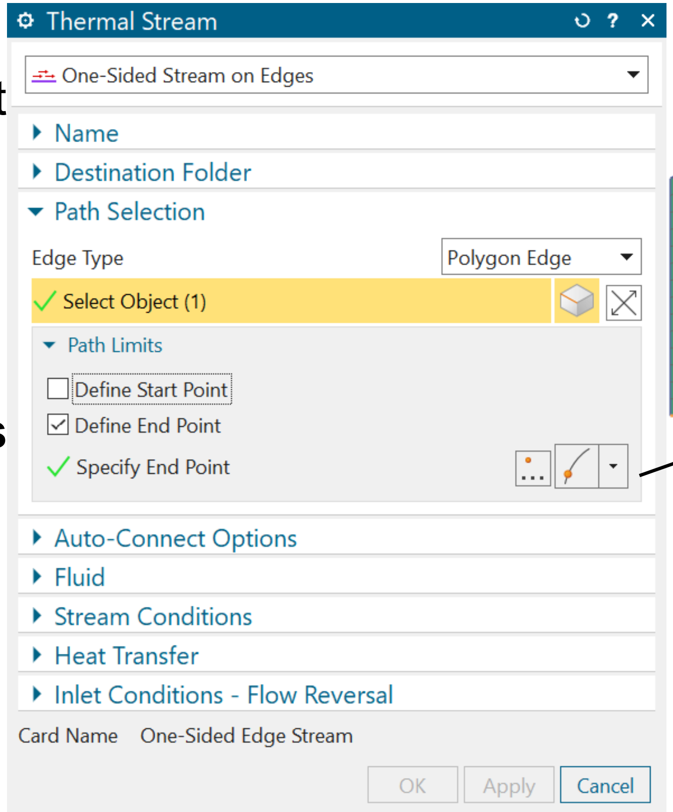
SIEMENS

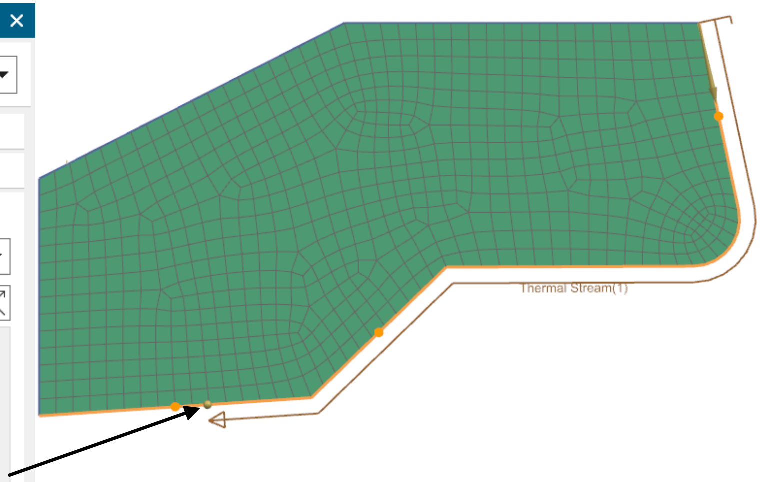
Thermal stream on edges

Use **Path Selection** by selecting “start point” and “end point” along surface to quickly select all edges in between. No need to divide or split CAD at these locations.

Tip: to prevent streams from jumping across multiple components, clear **Jump Edge Gaps** in the **Smart Selector Options** dialog box.





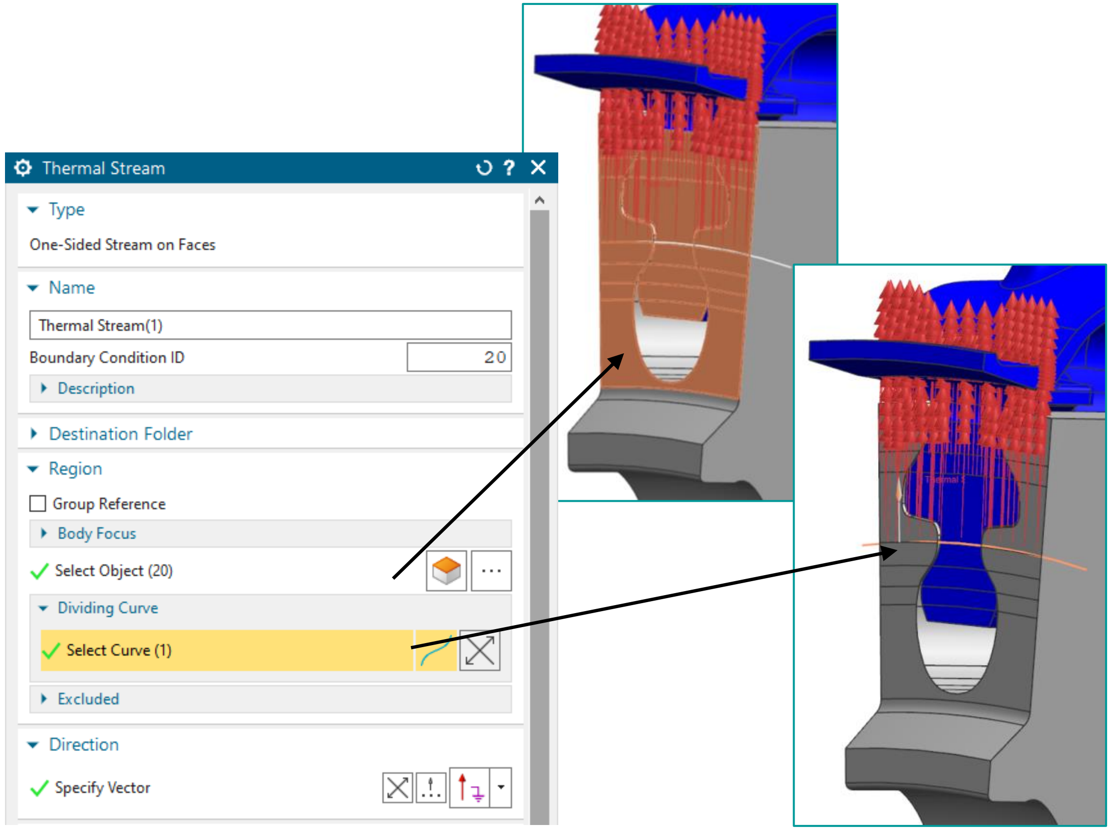


SIEMENS

Thermal stream on faces

- Define a vector to specify the flow direction. It does not have to be exact. As long as vector is not perpendicular to initial face, the flow direction will “follow” the curvature of the selected geometry.

- Use the **Dividing Curve** option to select a curve, applying the boundary condition only to the portion of the faces on one side of the dividing curve.
  - The curve must already exist.
    Use **Menu→Insert→Curves→Lines and Arcs** commands to create a curve.
  - It allows one curve per stream definition.



SIEMENS

Defining thermal stream condition

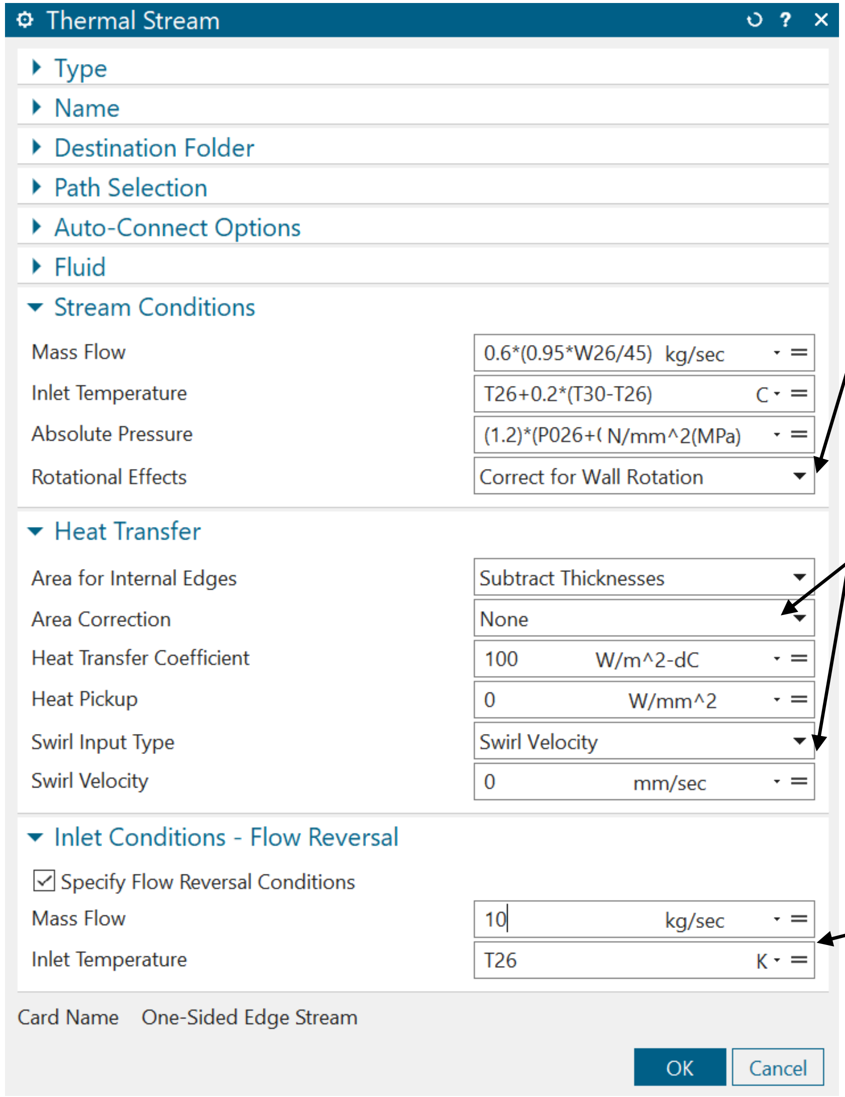
The energy equation for a one-sided thermal stream is:

$$\dot{m}C_p\frac{dT_f}{dt} = h\,A\left(T_f - T_s\right) + H_pA$$

- $\dot{m}$ is the specified mass flow rate of the stream.
- C_p is the specific heat of the stream fluid material.
- T_f is the fluid temperature, computed by thermal solver.
- T_s is the solid body temperature convecting with the stream, computed by the solver.
- A is the area of the solid body associated with the stream, computed by the solver.
- h is the specified heat transfer coefficient between the fluid and solid surface.
- H_p is the optionally specified heat pickup associated with the current element (per unit area).

You can define thermal streams with proprietary correlations, which are securely defined in a DLL file that is referenced in the **Custom Plugin** customer defaults.

Defining thermal streams



You can:

- Account for rotational effects. If **Rotation** load is applied and the **Correct for Wall Rotation** option is selected, the inlet temperature will be corrected for rotation based on input **Swirl Ratio** or **Swirl Velocity**.
- Specifies how the solver corrects the convective area. The convective area factor result set can help you investigate if you have set up your model correctly and the solver computes the values as expected.
- Specify flow reversal conditions.

SIEMENS

Modeling total temperature effects on wall due to rotating flow

In a system with rotating and stationary parts, the fluid temperature seen by a rotating surface and a stationary surface is different.

To account for total temperature effects, you need to consider both the static and dynamic components.

Fluid temperature types:

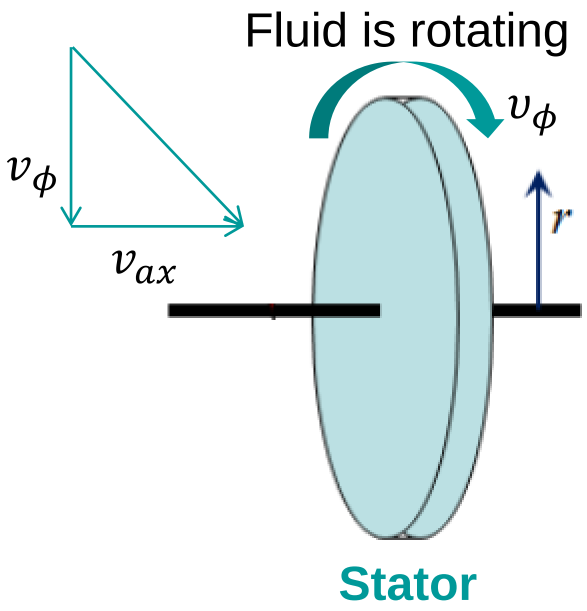
- Static temperature, T_s, is the temperature of a fluid that is not in motion. It is used to evaluate bulk fluid properties. Use the *GetFluidTemperatureOfType* plugin function to request the solver to output the temperatures if necessary.
- Inlet fluid temperature is used in convection calculations and can be:
  - Total Absolute Temperature, $T_{t,abs}$, is the total temperature relative to a stationary component.
  - Total Relative Temperature, $T_{t,rel}$, is the total temperature relative to a rotating component.

Using the Neglect Wall Rotation option to model total temperature effects

When you use the **Neglect Wall Rotation** option, the thermal solver calculates the total absolute fluid temperature, $T_{t,abs}$, to use for convection:

$$h_t\left(T_{t,abs}\right) = h_s(T_s) + \frac{(v_\phi^2 + v_{ax}^2)}{2}$$

$$\int^{T_{t,abs}} c_p(T)dT = \int^{T_s} c_p(T)dT + \frac{(v_\phi^2 + v_{ax}^2)}{2}$$



The fluid velocity is equal to 0, therefore $T_s = T_{t,abs}$.

- T_s, is the static temperature, temperature of a fluid that is not in motion.
- c_p is the specific heat of the stream fluid material.
- v_ϕ is the fluid swirl velocity.
- v_{ax} is the fluid axial velocity.

The thermal solver assumes that the specified <u>inlet fluid temperature</u> is <u>the total absolute fluid temperature</u>.

Use the **Neglect Wall Rotation** option when the components that surround the fluid are stationary or when the components are rotating at low speeds.

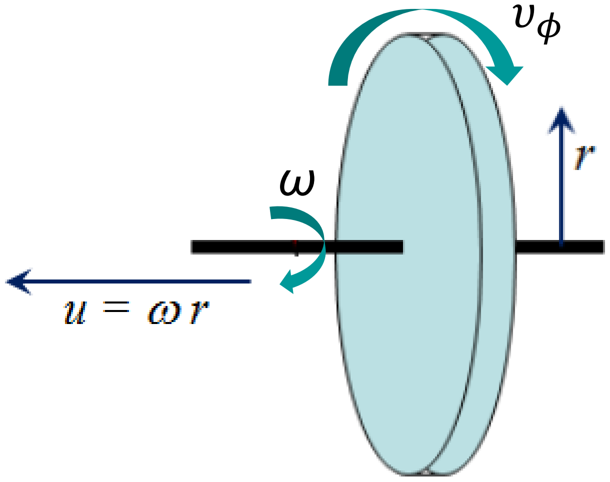
Modeling total temperature effects on wall due to rotating flow

Use the **Correct for Wall Rotation** option to calculate the total relative fluid temperature, $T_{t,rel}$, seen by the rotating components, which is determined from the total enthalpy:

$$h_t\left(T_{t,rel}\right) = h_s(T_s) + \frac{\left((u - v_\phi)^2\right) + v_{ax}^2)}{2}$$

$$\int^{T_{t,rel}} c_p(T)dT = \int^{T_s} c_p(T)dT + \frac{\left((u - v_\phi)^2\right) + v_{ax}^2)}{2}$$

Correct for Wall Rotation



Rotating disc

The total relative temperature is related to total absolute temperature as:

$$T_{t,rel} = T_{t,abs} + \Delta T_{rel}$$

- ΔT_{rel} is the relative temperature difference.
- v_ϕ is the fluid swirl velocity.
- v_{ax} is the fluid axial velocity, which is assumed to be 0.
- u is a wall velocity, where ω is the rotor rotational rate and *r* is the radius.

The thermal solver assumes that the specified inlet fluid temperature is the total absolute fluid temperature.

Use the **Correct for Wall Rotation** option when the components that surround the fluid are rotating.

Modeling total temperature effects on wall due to rotating flow

Use the **Relative Temperature Reference Frame** option when working in a relative reference frame to calculate the total relative fluid temperature, $T_{t,rel}$, using:

$$\int^{T_{t,rel}} c_p(T)dT = \int^{T_s} c_p(T)dT + \frac{((u - v_\phi)^2) + v_{ax}^2)}{2}$$

The total absolute fluid temperature is then recovered from the total relative fluid temperature:

$$\int^{T_{t,abs}} c_p(T)dT = \int^{T_s} c_p(T)dT + \frac{(v_\phi^2 + v_{ax}^2)}{2}$$

The thermal solver assumes that the specified inlet fluid temperature is the total relative fluid temperature.

You can define the angular velocity of this reference frame using the **Relative Reference Frame** type of the **Rotation** load. If you do not define the relative reference frame angular velocity, the thermal solver uses the maximum rotating speed of rotors as the reference frame angular velocity.

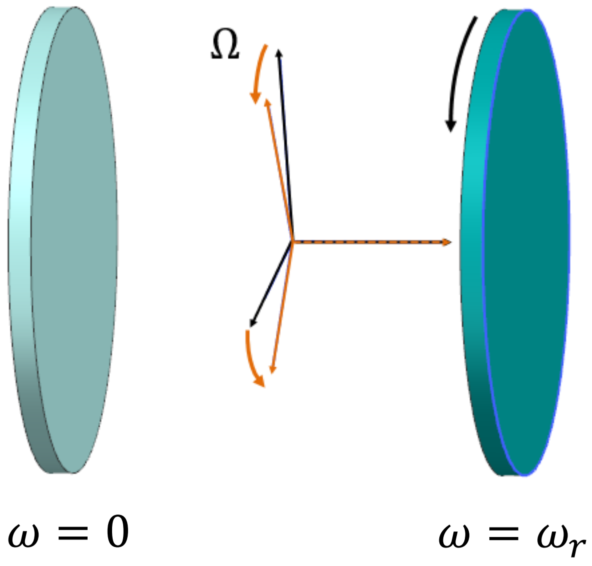
Use the **Relative Temperature Reference Frame** option when components that surround the fluid are rotating.

Understanding the relative reference frame

The thermal solver solves the heat transfer equations as a function of duct relative temperature, rotational speed of the wall, and swirl velocity in the specified relative reference frame.

You define the angular velocity of this reference frame using the **Relative Reference Frame** type of the **Rotation** load.

All 1D duct and stream elements in the 1D duct network reference the same relative reference frame for total temperatures when you set the **Rotational Effects** to **Relative Temperature Reference Frame** in defined thermal streams, and ducts coupling to the walls.



Relative reference frame (orange) between rotating and stationary parts

You define one relative reference frame for all 1D duct network that has a relative reference frame for total temperatures.

Accounting for rothalpy

When computing the total temperature effects in a relative reference frame, you can instruct the solver to conserve the rothalpy, I, the rotational stagnation enthalpy, by selecting the **Automatically Calculate Rothalpy** option.

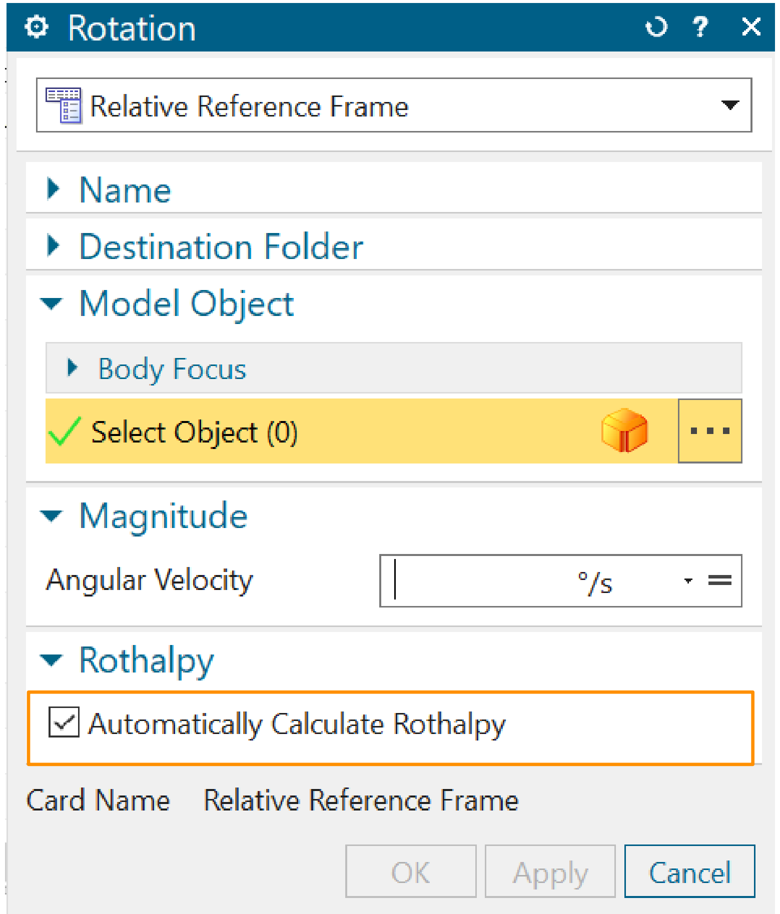
The thermal solver adds appropriate heat fluxes for windage and pumping due to rotation, when it computes the fluid total relative temperature.

The rothalpy along the radius is constant in adiabatic conditions:

$$I = \int^{T_{t,rel}} c_p(T)dT - \frac{r^2\Omega^2}{2} = constant$$

- Ω is the angular velocity of the reference frame.
- r is the fluid radius.

When this option is cleared, the thermal solver does not add the second term, $\frac{r^2\Omega^2}{2}$, to the energy equation to conserve the rothalpy.

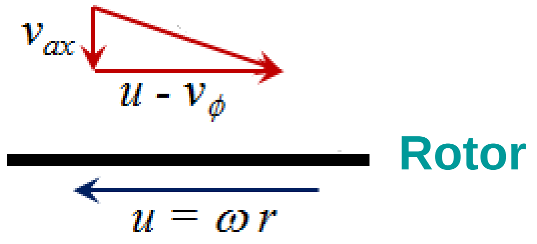


SIEMENS

Specifying swirl velocity

For all three methods of total temperature effects, you can specify the swirl velocity or the swirl ratio. When you specify the swirl ratio, the solver computes the swirl velocity as:

$$v_\phi = \chi_s \omega r$$



- v_ϕ is the fluid swirl velocity.
- $\chi_s = \omega_{fluid}/\omega$ is the fluid swirl ratio.
- u is a wall velocity, where ω is the rotor rotational rate and *r* is the radius.

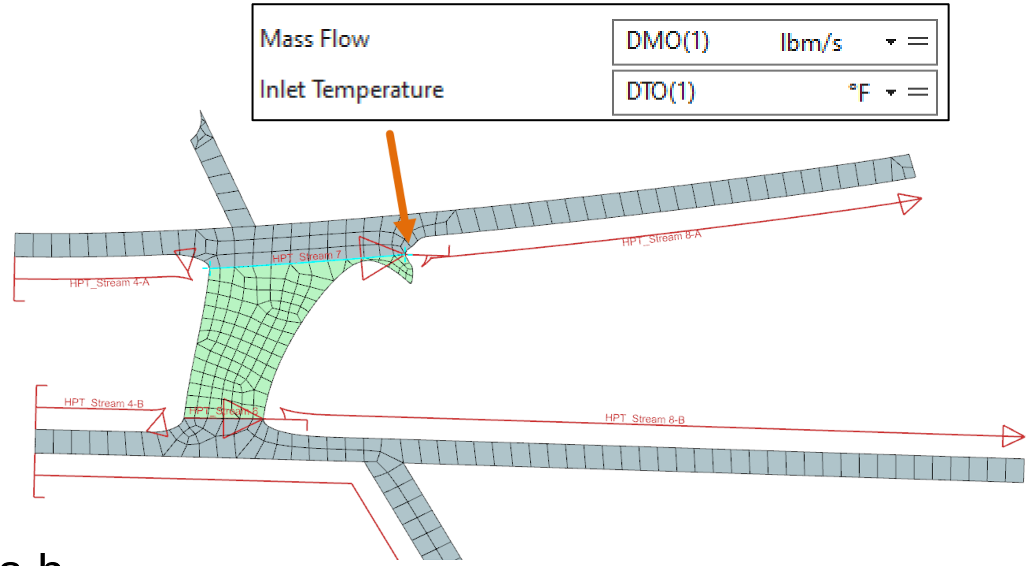
Connecting thermal streams

You can connect thermal streams by using:

- Built-in functions such as STO, SMO etc.
- The **Junction** simulation object to manually setup connections.
- Auto-connect options to connect streams that are geometrically in contact with each other.

Connecting thermal streams using the built-in functions

You can connect streams by specifying stream properties such as mass flow and temperature, coming from other streams using built-in functions.

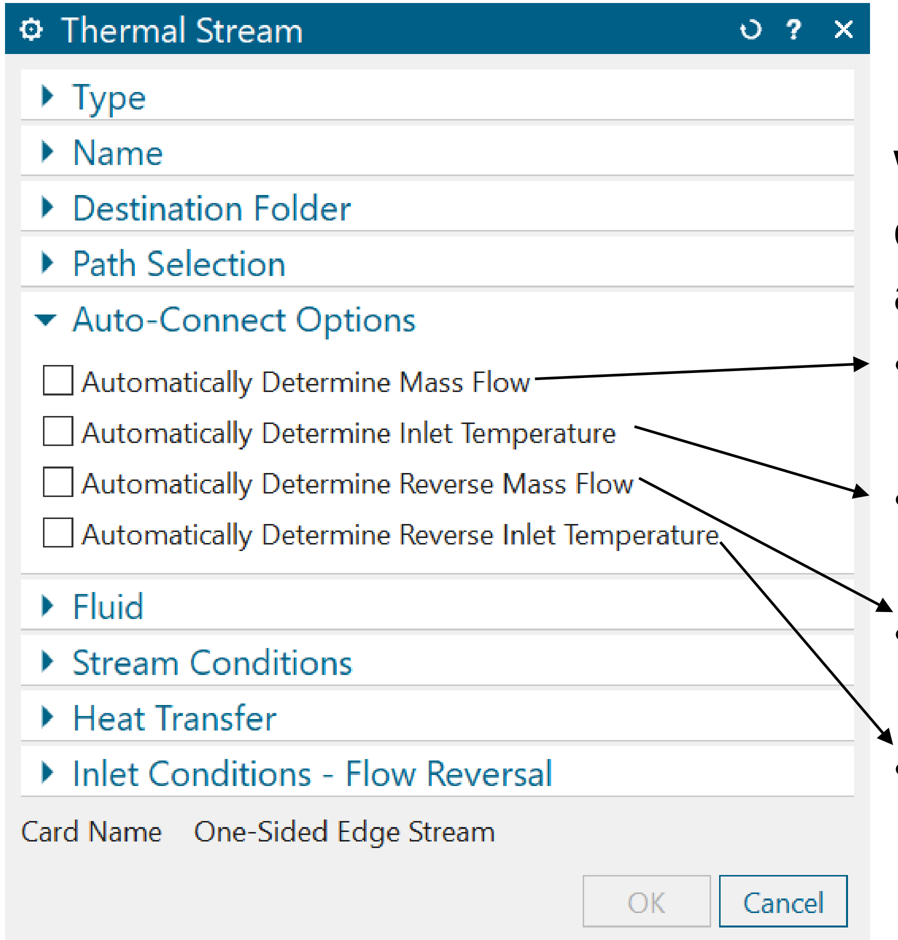


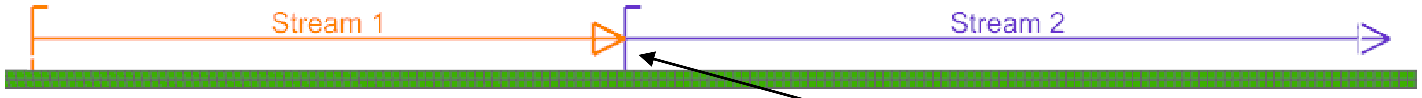
To specify stream properties, use:

- Thermal functions:
  - STO(i): outlet temperature of stream i.
  - SMO(i): outlet mass flow of stream i.
  - DTO(i): outlet temperature of Duct Label i.
  - DMO(i): outlet mass flow of Duct Label i.
  - MIX(a,b): mass-weighted average temperature of streams a,b.
  - MMIX(a,b): net mass flow of streams a,b.
  - SP(i): outlet pressure of stream i.
- Custom expressions or condition sequence parameters. For example: $T = 0.75(T3 - T25) + T25 + 18$, where T3 and T25 are condition sequence parameters.

SIEMENS

Connecting thermal streams using auto-connect options





User defined or natural CAD endpoint (vertex)

When thermal streams share an user defined or natural CAD endpoint or line, there are options to automatically ensure mass flow continuity and heat balance at the junctions.

- Mass flow is carried from upstream streams. It replaces DMO or MMIX function.
- Inlet temperature is taken from streams outlet. It replaces STO or MIX function.
- Mass flow is calculated based on upstream or downstream values if flow reverses.
- Inlet temperature is calculated based on upstream or downstream values if flow reverses.

Connecting thermal streams using auto-connect options

When a stream inlet or outlet has multiple geometrical and user-defined connections, the thermal solver merges all these connections to a single connection. Each connection provides one linear equation representing mass flow conservation at the connection.

$$\sum(m_{inlet}) = \sum(m_{leave})$$

The temperature at a connection depends on the temperature and mass flows of the fluid coming to the connection.

$$T_{mix} = \frac{\sum(H_{inlet})}{\sum(m_{inlet} C_{pinlet})}$$

where:

- H_i is the enthalpy of the inlet stream.
- m_i is the mass flow of the inlet stream.
- C_{pinlet} is the specific heat capacity.

SIEMENS

Connecting thermal streams using auto-connect options

There are several cases that the solver cannot resolve because the linear system of equations is either under-constrained or over-constrained.

Under-constrained settings

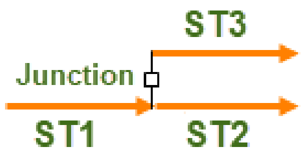
Thermal stream options		ST1	ST2
Mass flow	Automatically Determine Mass Flow check box	[ ]	[x]
	Mass Flow box	Specified: 0.1 kg/s	Not specified
Reverse mass flow	Automatically Determine Reverse Mass Flow check box	[x]	[ ]
	Reverse Mass Flow box	Not specified	Specified: 0.1 kg/s

Correctly specified settings

Thermal stream options		ST1	ST2
Mass flow	Automatically Determine Mass Flow check box	[ ]	[x]
	Mass Flow box	Specified: 0.1 kg/s	Not specified
Reverse mass flow	Automatically Determine Reverse Mass Flow check box	[ ]	[x]
	Reverse Mass Flow box	Specified: 0.1 kg/s	Not specified

The thermal solver computes mass flow balance of all streams that are connected geometrically and through a junction.

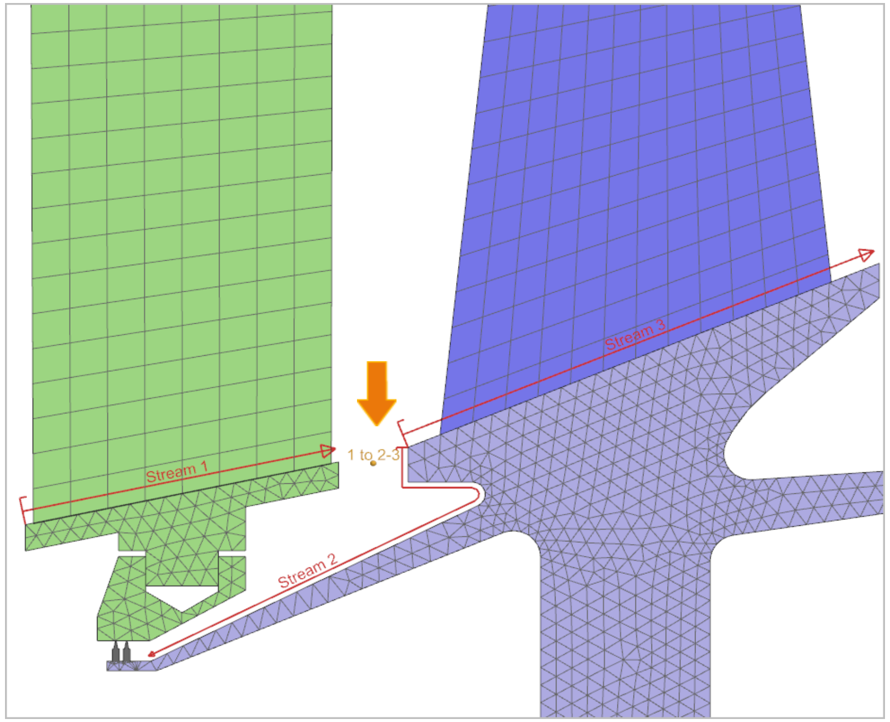
Thermal stream options		ST1	ST2	ST3
Mass flow	**Automatically Determine Mass Flow** check box	[ ]	[ ]	[x]
	Mass Flow box	Specified: 0.1 kg/s	Specified: 0.05 kg/s	Not specified



Connecting thermal streams using Junction

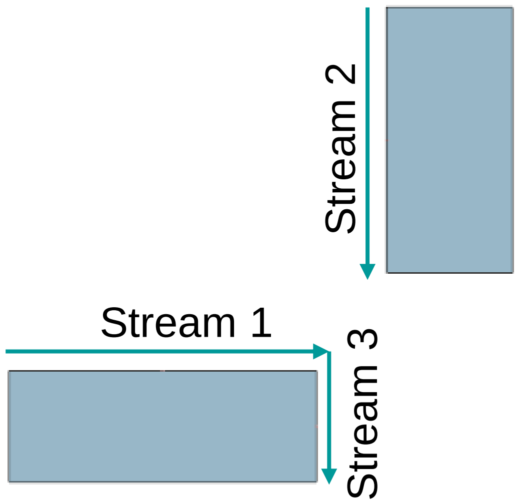
When streams do not share a common endpoint or edge, you can use the **Junction** command to manually connect streams at a junction for streams that are disconnected in the digital model, but are connected in the physical model to ensure mass flow continuity and heat balance.

- It allows you to specify incoming and outgoing streams at a junction.
- It simplifies setup of stream connections.
- Together with the **Auto Connect** option in thermal streams, the solver computes the mass flow and properties at the junction.



Connecting thermal streams using Junction and auto-connect options

In the following example, the auto-connect option is enabled for Stream 3, establishing a connection with Stream 1. Additionally, Stream 3 is linked to Stream 2 using a **Junction** command.

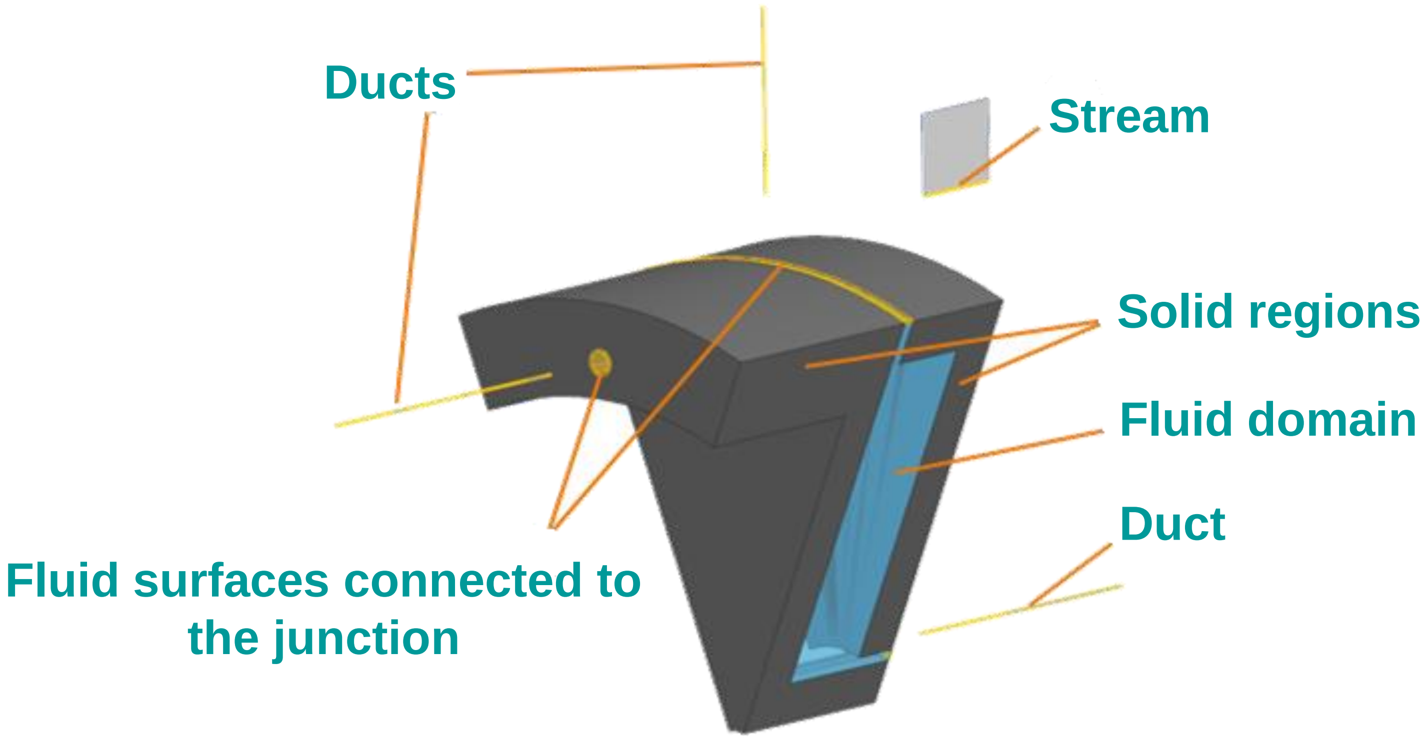


The thermal solver calculates the inlet temperature for the Stream 3 as:

$$T_{mix} = \frac{(m_2 T_2 + m_1 T_1)}{m_2 + m_1}$$

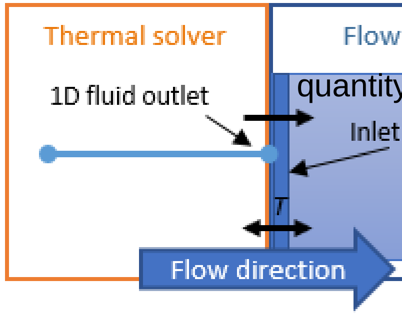
Connecting ducts and streams to 3D fluid domain

Use the **Ducts/Streams Junction to 3D Flow** type of the **Junction** simulation object to connect 3D CFD fluid domain to streams and ducts from 1D fluid network. You choose one of the following quantities to be transferred to the fluid domain from the thermal streams and ducts: mass flow or pressure.



Connecting ducts and streams to 3D fluid domain

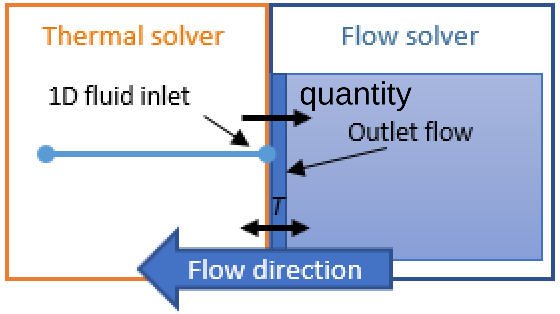
Fluid flows from 1D network to 3D domain



Transferring mass flow: The flow solver treats the selected 2D surface as an inlet flow.

Transferring pressure: The 1D fluid pressure from the outlet of the stream or duct is specified as the opening total pressure.

Fluid flows from 3D domain to 1D network



Transferring mass flow: The flow solver treats the selected 2D surface as an outlet flow.

Transferring pressure: The 1D fluid pressure from the inlet of the stream or duct is specified as the opening static pressure.

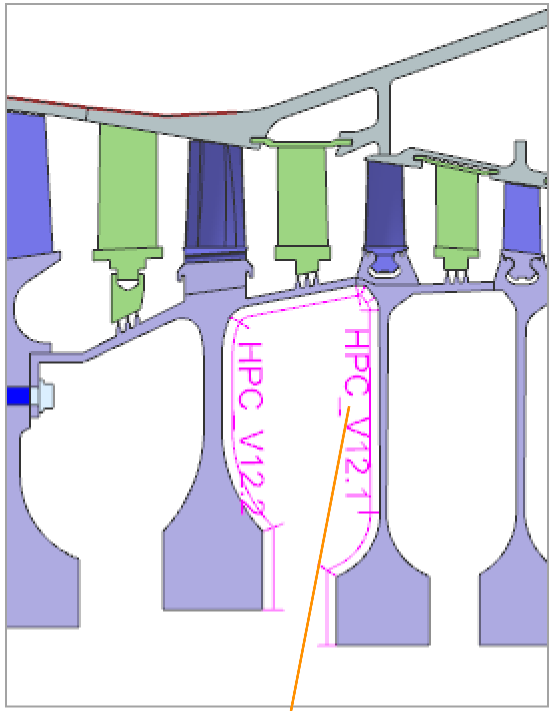
SIEMENS

Defining thermal voids

Use the **Thermal Voids** load to model convection in cavities or chambers where the temperature of the fluid is influenced by multiple regions. All surfaces in the void will convect to the same fluid temperature which is calculated during the solve.

You can define:

- A single **Thermal Void** load that references up to 20 void regions (surface or edge) which define the pressure and heat transfer characteristics of each region.
- The area correction for internal edges that have different thicknesses on either side.



Thermal Void on the cavity fluid region of an axisymmetric 2D model

Defining thermal void conditions

To define a thermal void, you must specify heat load and capacitance in the energy equation for void:

$$\rho V C_p \frac{dT_f}{dt} = h\,A\left(T_f - T_s\right) + Q$$

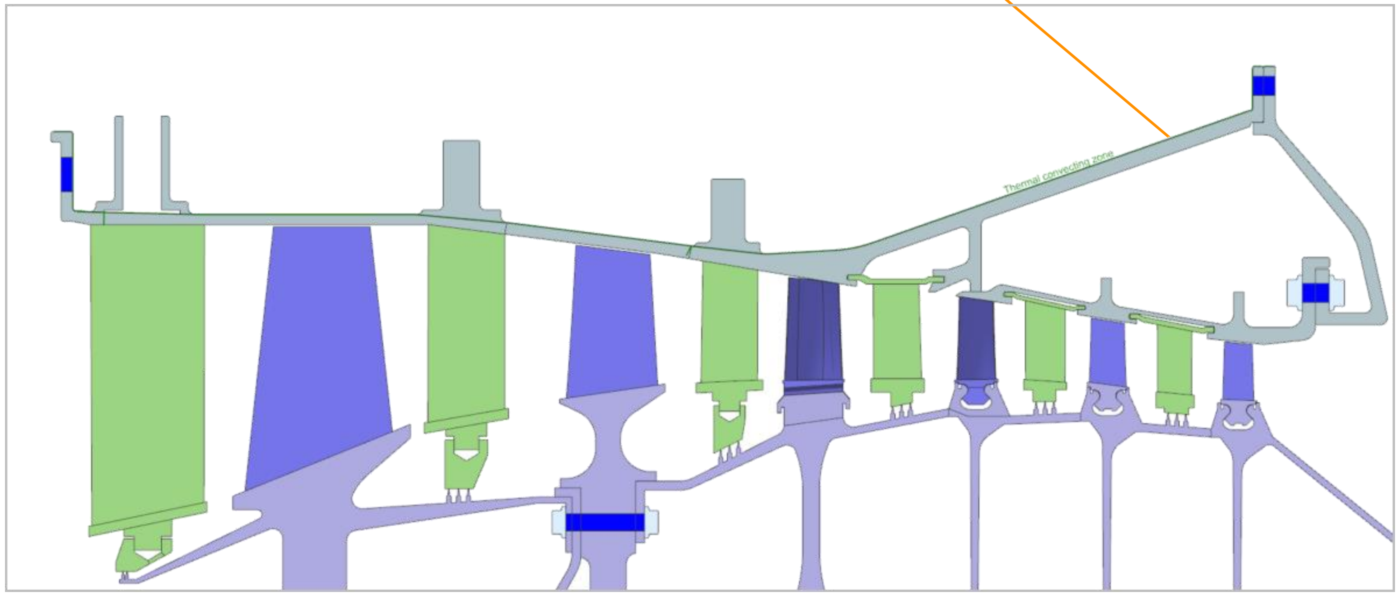
- V is the volume of the void cavity.
- ρ is the density of the fluid material in the void cavity.
- C_p is the specific heat of the fluid material in the void cavity.
- T_f is the void fluid temperature, computed by thermal solver.
- T_s is the solid body temperature, computed by the solver.
- A is the solid-fluid interface area determined by the solver.
- h is the specified local heat transfer coefficient.
- Q the specified heat load into the void in Watts. If a stream is feeding into a void, the PWR standard function may be used. If a void is feeding another void, the PWRV function can be used.

Defining thermal convecting zones

Use the **Thermal Convecting Zone** load to define the heat transfer for a part of the model that is convecting to a fluid at a known temperature. You can:

- Define the pressure at the model surface.
- Define heat transfer characteristics.
- Spatial and time-varying convection coefficient.

Thermal Convecting Zone on the outer surface of the casing



The governing equation:

$$Q = hA(T_f - T_m)$$

- Q is the heat flux from the metal to the prescribed temperature fluid volume, which can vary with time and space.

SIEMENS

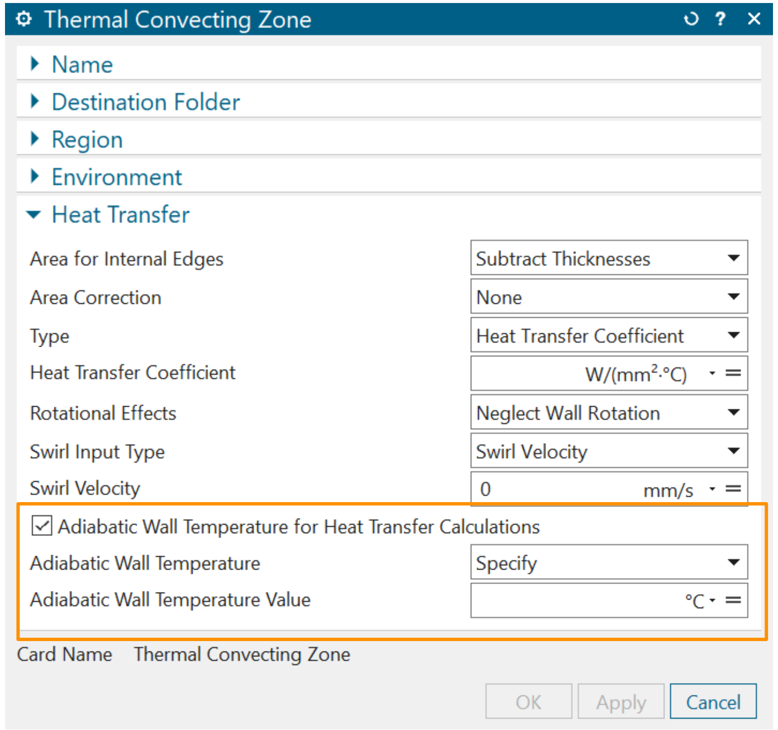
Considering adiabatic wall temperature for heat transfer

Use the adiabatic wall temperature instead of the fluid temperature for heat transfer calculations in the **Thermal Convecting Zone** load to obtain accurate convective wall heat fluxes for flows with significant viscous heating such as high speed flows in rotating and stationary machinery. The thermal solver computes the wall heat flux as:

$$q_w = h(T_w - T_{aw})$$

$$T_{aw} = T_s + RF\frac{v_{rel}^2}{2C_p}$$

- h is the heat transfer coefficient.
- T_w is the wall temperature.
- T_{aw} is the adiabatic wall temperature.
- $v_{rel} = abs(u - v_\phi)$ is the relative tangential velocity, where u is the wall tangential velocity and v_ϕ is the swirl velocity.
- C_p is the specific heat of the fluid material in the void cavity.
- $RF = Pr_{film}^{1/3}$ is the recovery factor, where the Prandtl number calculated at $T_{film} = \frac{1}{2}(T_w - T_f)$



The **Adiabatic Wall Temperature for Heat Transfer Calculations** option is also available in **Thermal Coupling – Convection** and **Duct Node Convection Coupling.**

SIEMENS

Boundary condition ID in expressions

When you create streams, voids, and convecting zones, and duct labels the software assigns a numerical ID to the boundary condition.

To change the numerical ID of these boundary conditions, you can:

- Edit the numerical ID in the dialog box for the boundary condition.
- Use the **Boundary Condition ID Manager** command. Use the **Renumber** and **Offset** options for changing ID numbers.

The following examples show how the numbering of the streams is changed depending on selected options.

If you select the **Renumber** option, and enter 11 in the **From** box, the stream ID numbers change as follows:

- Stream 3 → Stream 11
- Stream 1 → Stream 12
- Stream 4 → Stream 13
- Stream 2 → Stream 14

If you select the **Offset** option, and enter 11 in the **By** box, the stream ID numbers change as follows:

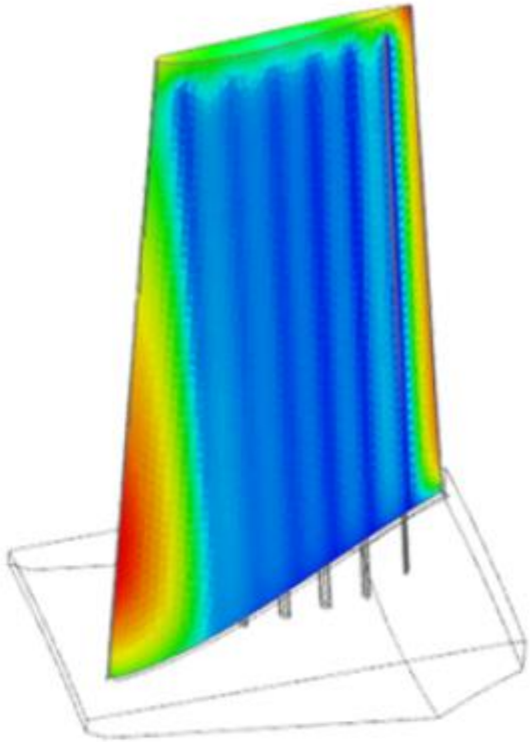
- Stream 3 → Stream 14
- Stream 1 → Stream 12
- Stream 4 → Stream 15
- Stream 2 → Stream 13

Modeling thermal barrier coating

Use the **Protective Layers** command to add thermal barrier coatings on top of thermal elements for protective layer heat transfer modeling and account for added mass distribution in a structural modeling.

You can:

- Create multiple protective layers.
- Assign different material properties for each layer.
- Vary thickness of each layer in a spatial field.
- Apply thermal boundary conditions on layers.
- Post process contour plots at the top, bottom, and middle of any layer.

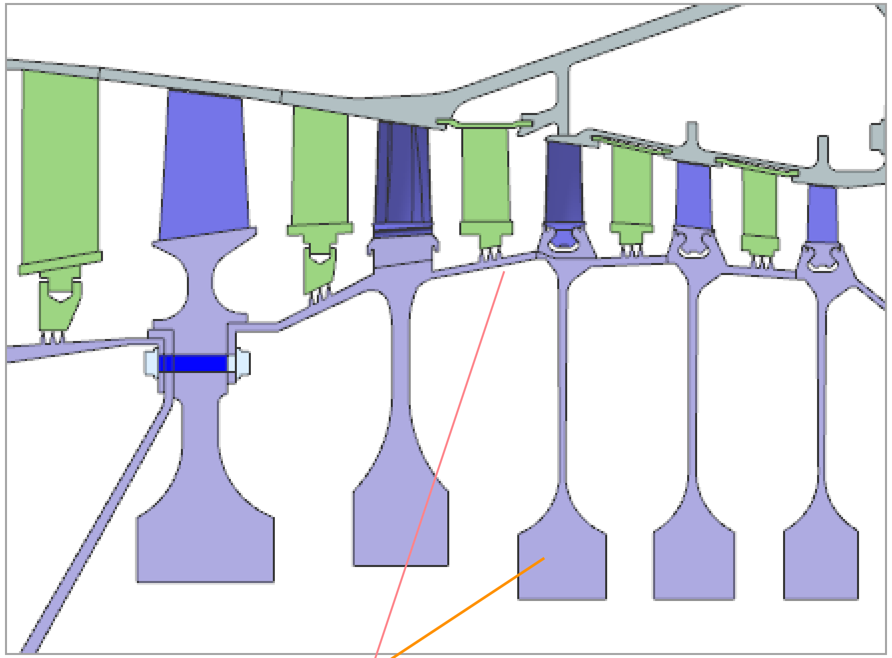


Modeling radiation effect inside aeroengine

Use the **Radiation** command to create view factor calculation requests for enclosures comprised of selected geometry or elements inside the aeroengine. The thermal solver computes radiative conductance between elements associated with the selected geometry.

You can define:

- **Enclosure Radiation**
- **All Radiation**



Radiation between the cavity surfaces

Defining structural load

You can define:

- **Rotation** load to model the rotating movement for the rotating parts of the model using, for instance, the global cyclic analysis coordinate system.
- **Bolt Pre-Load** to apply a pre-load to a bolt modeled with 2D plane stress and axisymmetric elements.
  - Force: Load on cut-plane method.
  - Displacement: Uniform strain method.

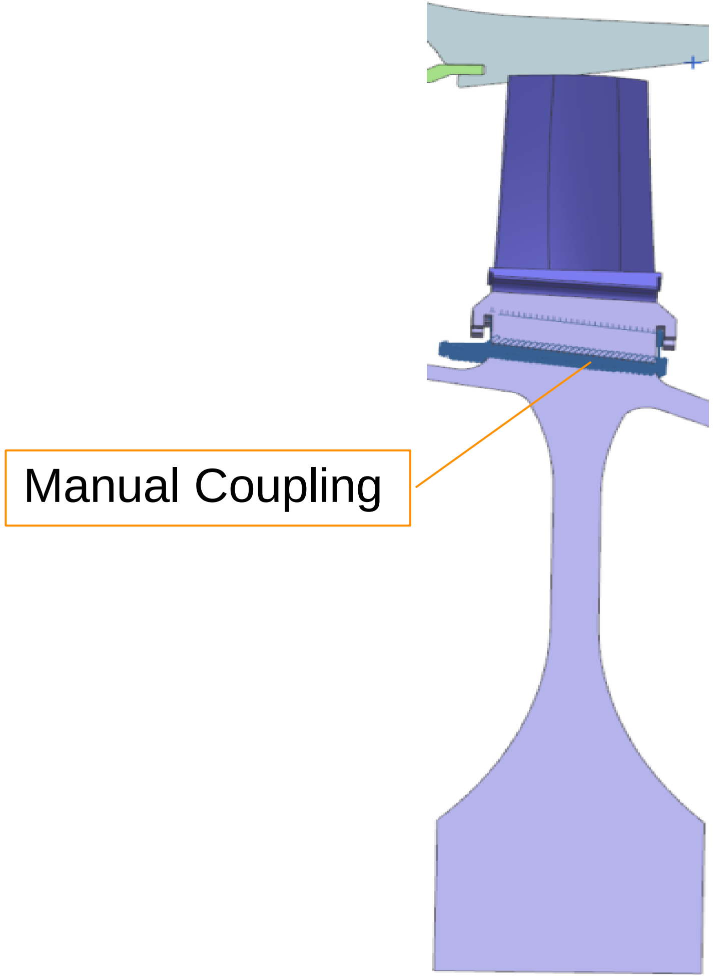
    The Force method is preferred. It is more efficient, no need to manually iterate on the axial strain, the solver iterates. However, if nodes on the cut plane are not coplanar, use the Displacement method.
- **Component Force XY** type of the **Force** load to define, for example, wire seal load, torque induced axial/radial separation, or gas path loads on airfoils.
  - Loads are defined with the **Condition Sequence Parameters**.
  - Each solution defines which **Condition Sequence** is used for this load definition.

Defining structural constraints

You can use:

- **User Defined Constraint** to constrain the tangential movement of the 3D solid bodies of the model to remove rigid body movement and axially constrains the rotor and the stator for a structural analysis.
  You can define constraints for up to six degrees of freedom (DOF). The DOF correspond to the nodal displacement coordinate system. Each DOF can be set to fixed, free, or a displacement value.

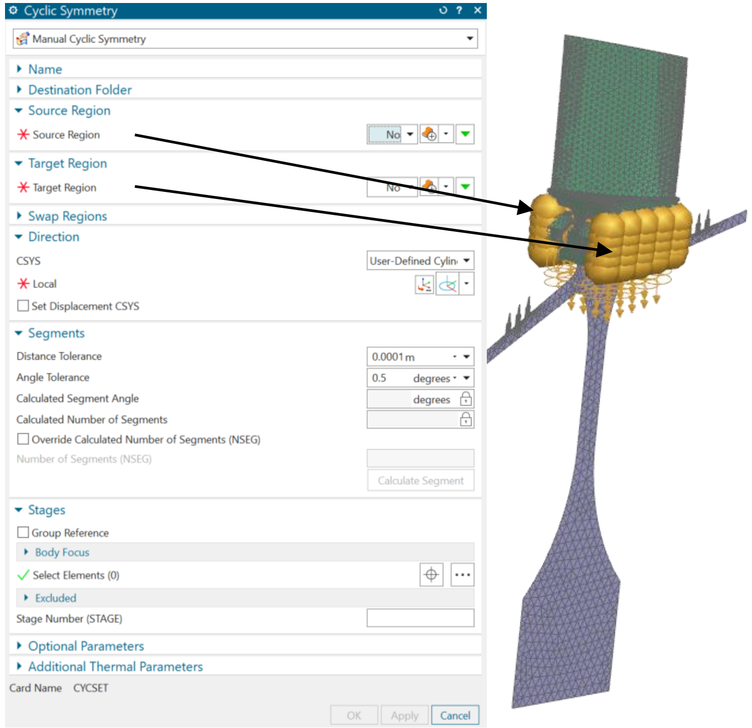
- **Manual Coupling** to define the coupled structural degrees-of-freedom (DOF) between selected nodes, for example, a 2D rotor disk and 3D rotor disk. Best practice is to use polygon geometry associated with nodes.



Using cyclic symmetry for material properties and area scaling

Use a **Cyclic Symmetry** simulation object to scale the material properties, and convective areas to account for the number of 3D segments in the model.

- Select source and target regions.
  - When using cyclic symmetry for a 2D/3D model you must exclude face at 2D/3D interface. Otherwise model would be over-constrained with cyclic boundary condition and nodal coupling.
- Select a method for defining the number of segments. Click **Calculate Segment** and verify **Number of Segments** matches modeling intent. If error is returned, click **Swap Regions**.
- Select the global cylindrical coordinate system if you did not define it in the FEM file. The coordinate system you select must be cylindrical and its Z-axis must be collinear with the global cyclic analysis coordinate system.
- In the **Stages** group, select the elements on the mesh that you want to include in this stage and set the stage number for the selected elements to have appropriate area and mass scaling.



SIEMENS

Structural and thermal deactivation sets

- Removes all selected elements from any calculation by the solvers.
- Can select elements, meshes, polygon bodies, curves such as fluid network and points such as 0D mass points.

Structural Deactivation Set:

- Ignores the deactivated elements while performing the structural calculations.
- No deflection or contact status updates will be returned to the thermal solver.

Thermal Deactivation Set:

- Ignores the deactivated elements while performing the thermal calculations.
- No body temperature or surface pressure will be passed to the structural solver.

Note:

- If thermal set deactivates a portion of the model used for the structural solution, model will fail model setup check. You need to apply temperature load to portion of structural solution with thermal deactivation set to solve.
- Coupled Thermal-Structural solution or Thermal solution will not run if 0D elements (mass points) are not deactivated for the thermal solve.